ANSYS®

Realize Your Product Promise®

2019 R1

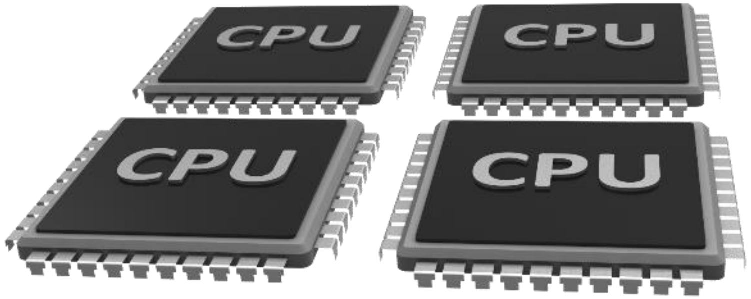
Mechanical Products Update – January 2019

AGENDA

- SOLVERS & HPC
- CONTACT
- FRACTURE
- MECHANICAL
- CMS
- RBD
- LINEAR DYNAMICS
- ACOUSTICS
- EXPLICIT DYNAMICS
- LS-DYNA
- EXTERNAL MODEL
- GENERAL AXISYMMETRIC
- MAPDL ELEMENTS
- MATERIAL DESIGNER
- TOPLOGY OPTIMIZATION
- LEVEL SET
- ADDITIVE MANUFACTURING
- COMPOSITES
- AQWA
- CLOUD

Mechanical APDL
2019 R1 Release

Distributed ANSYS Enhancements



- **New features**
  - Now default for all simulations!
  - Support for SMART fracture
  - Support for prestressed substructuring generation pass
  - Support for substructuring generation pass restarts
- **Improved scaling**
  - Significantly improved scaling when contact is present

DMP
- Distributed Memory Parallel

SMP
- Shared Memory Parallel

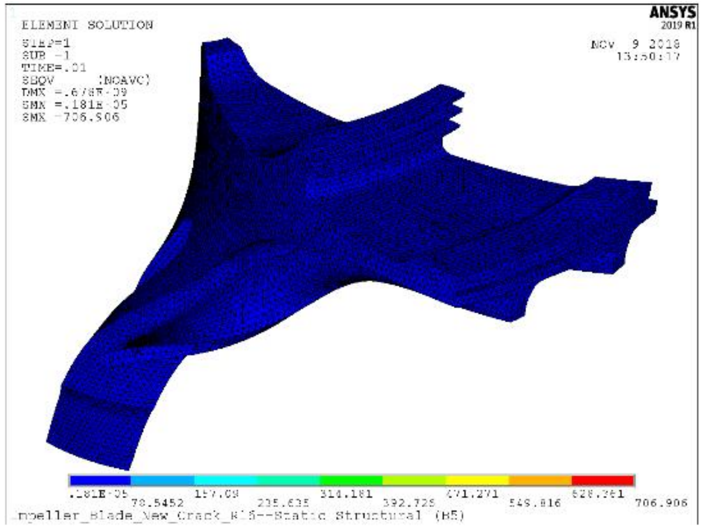
Distributed ANSYS Enhancements

- **Distributed ANSYS is now default**
  - Distributed memory parallel (DMP) is now enabled by default when MADPL is launched ("-dis" command line argument no longer necessary)
  - To revert back to shared memory parallel (SMP) use "-smp" command line option (or select SMP via the MAPDL launcher)

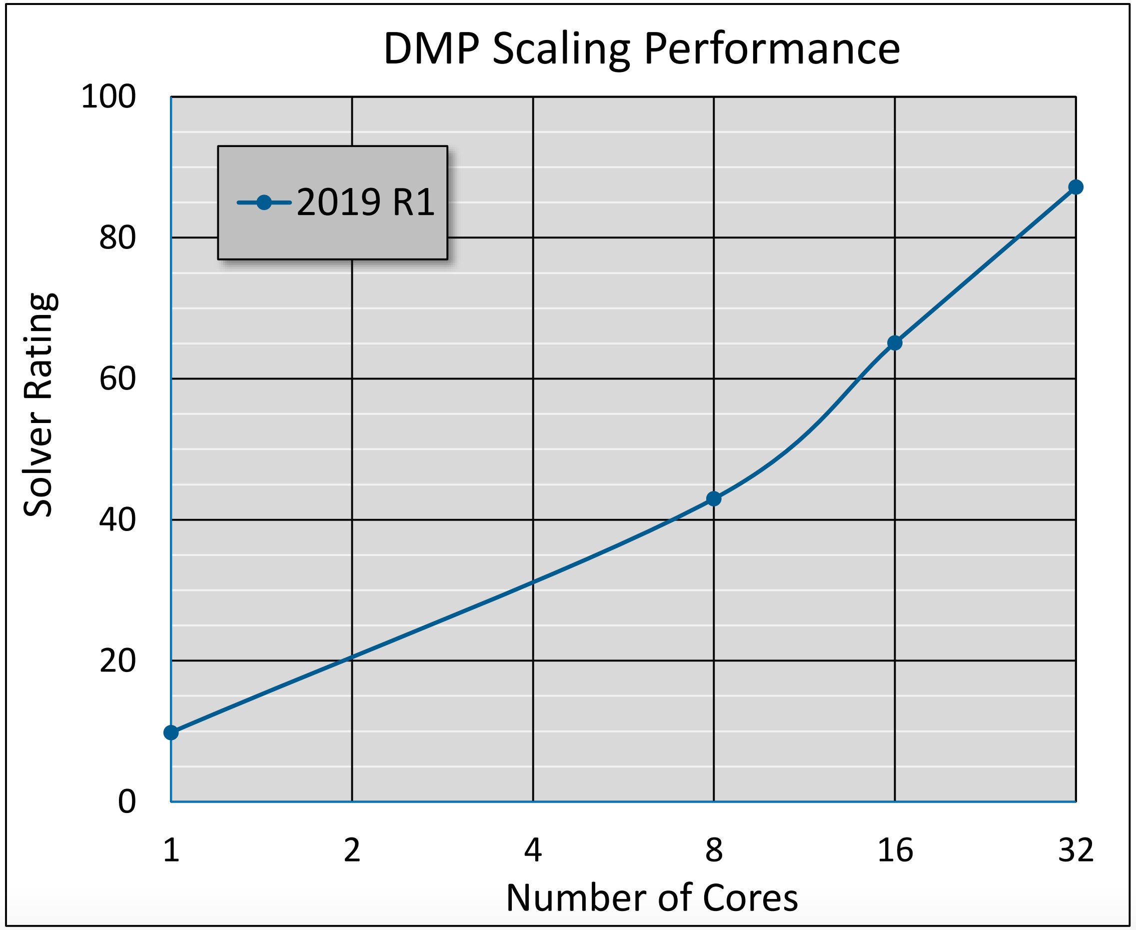
Caution:
Operations that were not supported in DMP in previous releases have NOT been automatically parallelized in this release

Distributed ANSYS Enhancements

- **Support added for SMART Fracture**



- 2.7 million DOF; sparse solver
- Nonlinear static analysis involving fracture with 101 crack tips
- Linux server; 4 Intel Xeon E7-8867v3 processors, 512GB RAM, SSD, RHEL 7.0



Distributed ANSYS Enhancements

- **Scaling when contact is present**
  – Some indicators for when contact is a performance bottleneck

```
 D I S T R I B U T E D   D O M A I N   D E C O M P O S E R

  ...Number of elements: 635691
  ...Number of nodes:    628272
  ...Decompose to 256 CPU domains
  ...Element load balance ratio =      7.588
```

Values > 4 generally indicate serious workload imbalance

```
 *** WARNING ***                         CP =      291.883   TIME= 01:04:10
The work load is highly imbalanced for this solution, likely due to the
contact pair(s) in the model.  This imbalance will negatively impact
the scaling of Distributed ANSYS.  Please review the contact pair
associated with real constant set 55 to see if it can be altered to
help improve performance.  See the Performance Guide for more details
on scaling and for some tips on how to manually alter the contact
definition to improve the work load balance.
```

Check for this warning

Distributed ANSYS Enhancements

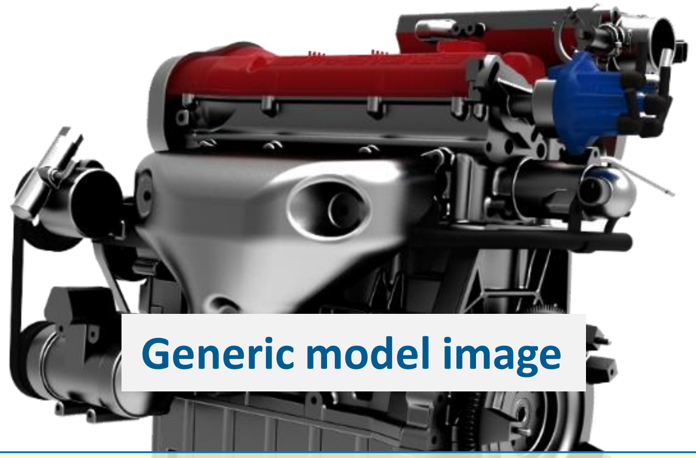
- **Scaling when contact is present**
  - <u>Symmetric contact pairs improved</u>
    - In previous releases both sets of contact and target surfaces for a symmetric contact pair would reside in a single domain → poor load balance
  - <u>MPC contact pairs improved</u>
    - In previous releases if an MPC and a non-MPC contact pair touched or overlapped, they would both reside in a single domain → poor load balance
  - Goal → Reduce the load imbalance

Distributed ANSYS Enhancements

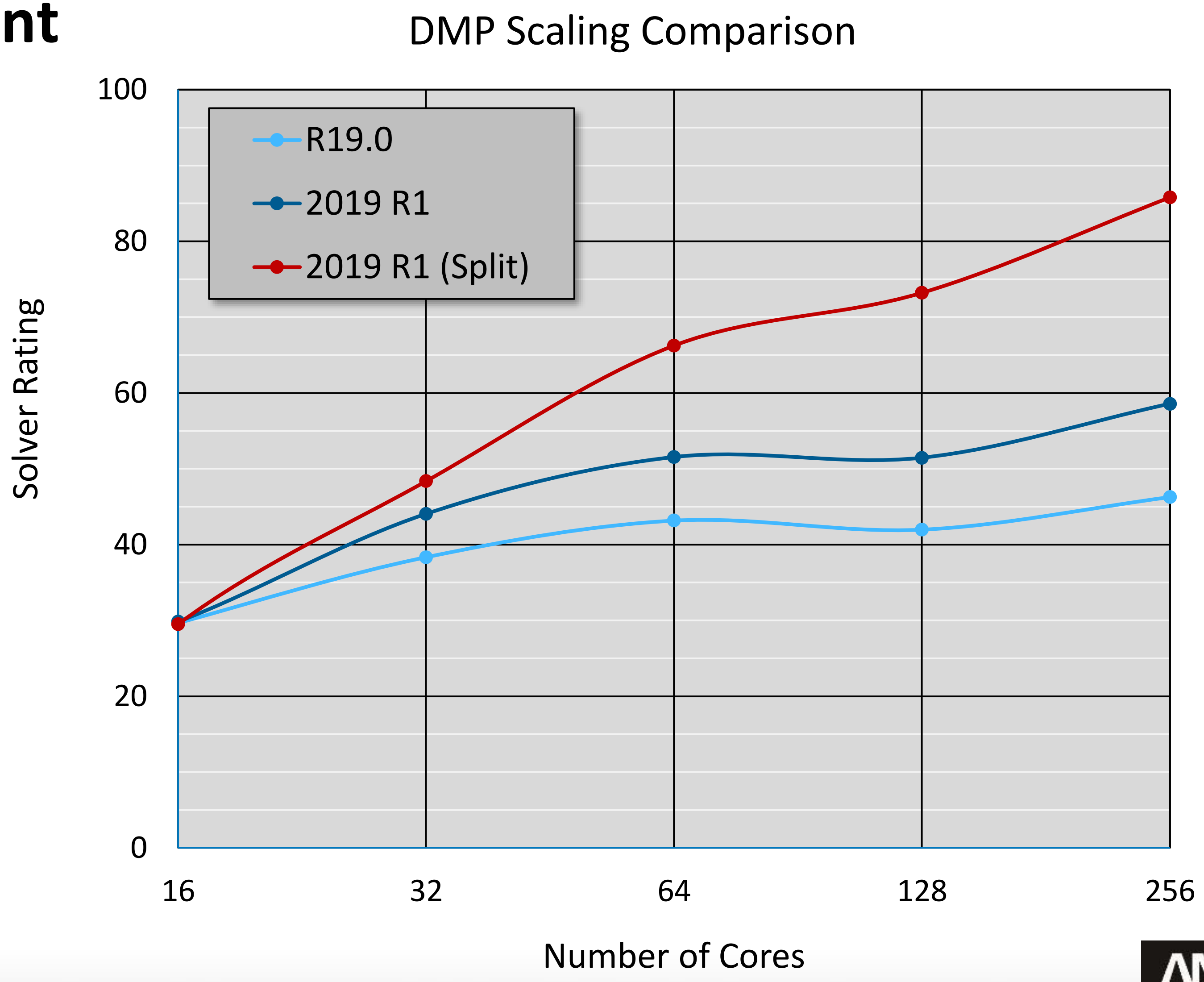
- **Contact pair splitting**
  - CNCH,DMP → Automatically split contact pairs during solution
  - CNCH,SPLIT → Manually split contact pairs
  - CNCH,MERGE → Manually merge contact pairs

Distributed ANSYS Enhancements

- CNCH,DMP scaling improvement

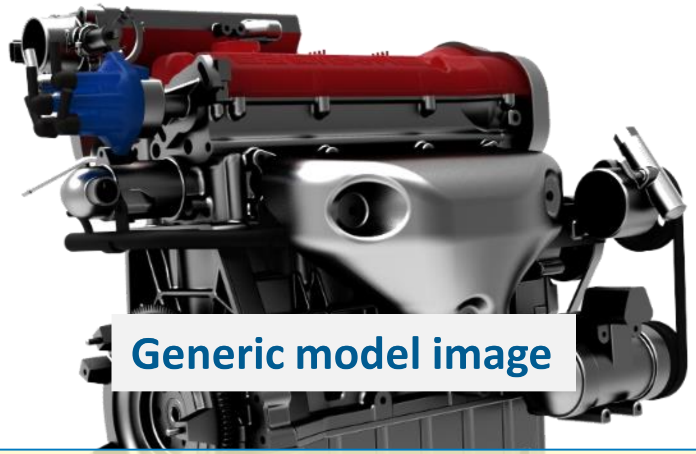


- 9.1 million DOF; sparse solver
- Nonlinear static analysis involving contact, plasticity and gasket elements
- Linux cluster; each compute node contains 2 Intel Xeon E5-2695v3 processors, 256GB RAM, SSD, CentOS 7.2
- Mellanox FDR Infiniband

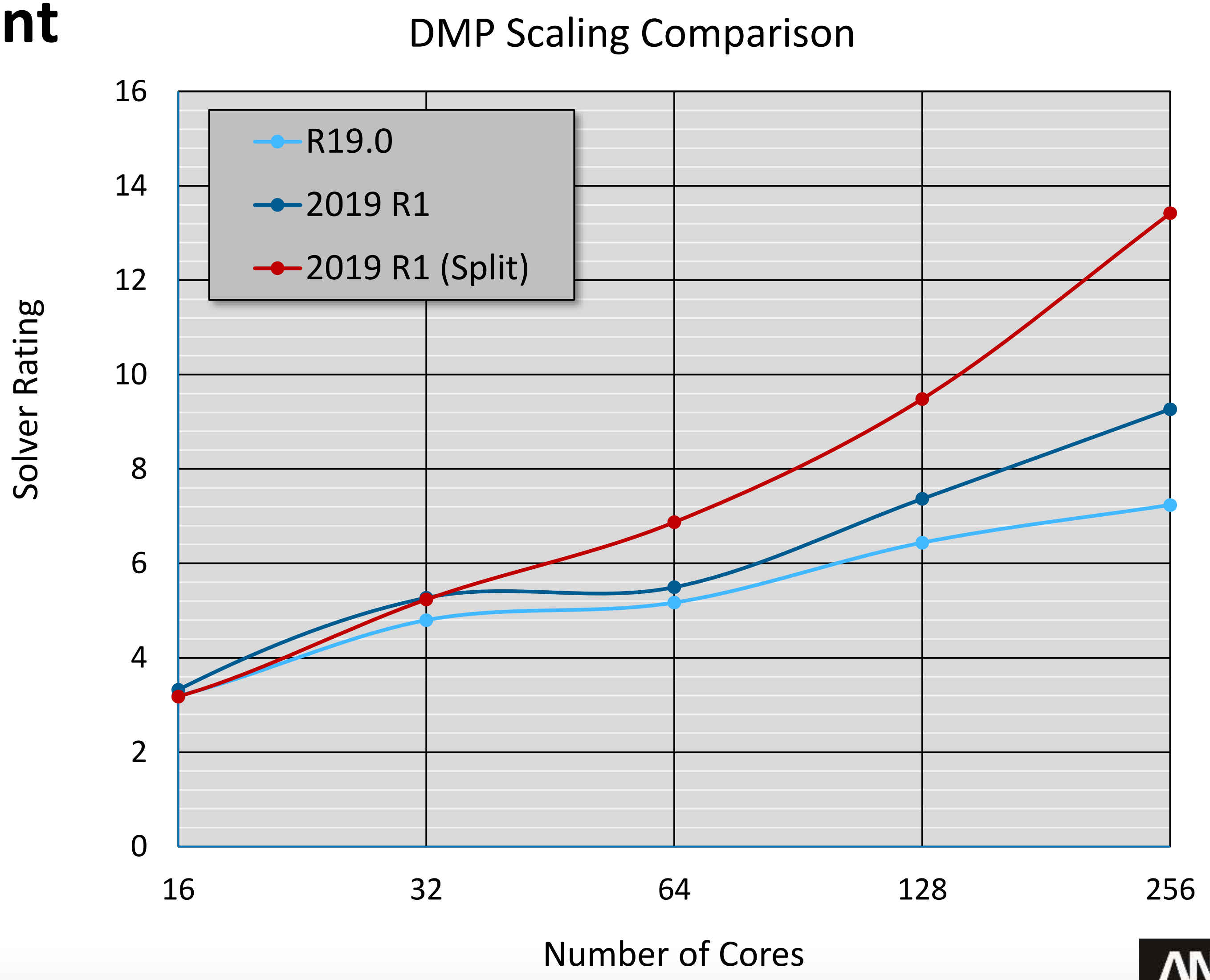


Distributed ANSYS Enhancements

- **CNCH,DMP scaling improvement**



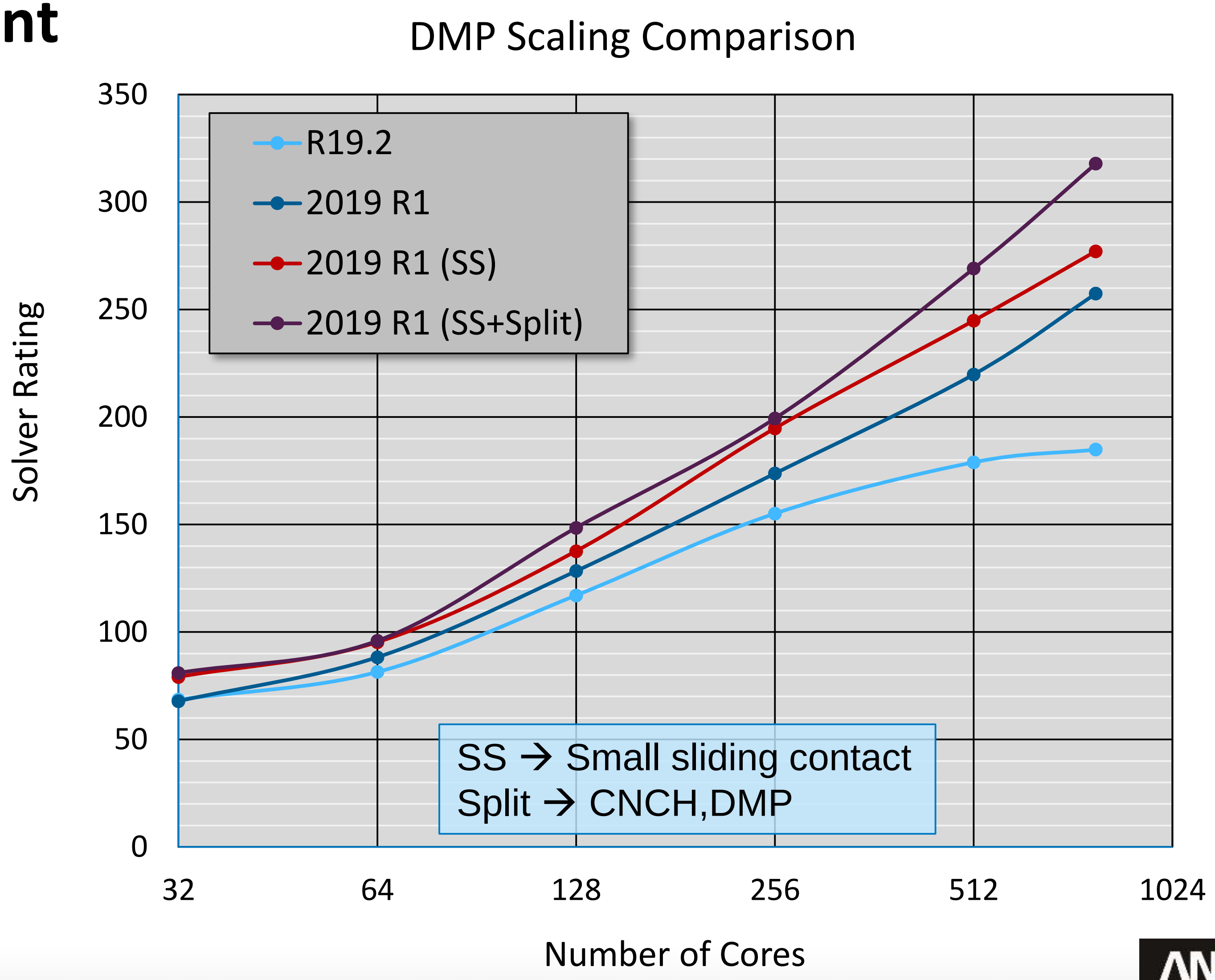
- 13.4 million DOF; sparse solver
- Nonlinear static analysis involving contact, plasticity and NLGEOM
- Linux cluster; each compute node contains 2 Intel Xeon E5-2695v3 processors, 256GB RAM, SSD, CentOS 7.2
- Mellanox FDR Infiniband



Distributed ANSYS Enhancements

- **CNCH,DMP scaling improvement**

- 10.2 million DOF; sparse solver
- Nonlinear static analysis involving contact, plasticity
- Linux cluster; each compute node contains 2 Intel Xeon Gold 6148 processors, 384GB RAM, SSD, CentOS 7.3
- Mellanox EDR Infiniband



Results File Enhancements

- **OUTRES command changes**
  - Eight new result item labels have been added

- **New compression algorithm available (sparsify)**
  - Activated via the /FCOMP,RST,SPARSE command
  - Lossless compression of results file data
  - Typically reduces results file size by 10-50%
  - Virtually no performance penalty

Results File Enhancements

- **OUTRES command changes**

OUTRES,Item,Freq,Cname,--,NSVAR,DSUBres

New labels added to provide additional user control

Results File Enhancements

- **OUTRES command changes**
  - New labels enable additional user control over results file data

ETMP	Element temperatures
CONT	Element contact data
NLDAT	Element nonlinear data
EHEAT	Element heat generation rate
FMAG	Electromagnetic nodal forces
CURD	Element source current densities
BKSTR	Element back stresses (requires EPPL too)
SRFS	Element surface stresses

Results File Enhancements

- OUTRES command changes

General Labels

ALL, BASIC, STAT, ERASE

Nodal quantities

NSOL, RSOL, V, A

CINT

Element quantities

ESOL

NLOAD, STRS, EPEL, FFLUX, FGRAD

EPPL, EPTH, EPCR, VENG, MISC

EPDI, AESOL, LOCI, SVAR

FMAG, ETMP, EHEAT, SRFS

NLDAT, CONT, BKSTR, CURD

New in 2019 R1

Results File Enhancements

- **OUTRES command changes**
  - –MAPDL default
    - OUTRES,ALL → write everything except LOCI, SVAR, V, A
    - No change in 2019 R1
  - –WB/Mechanical default

R19.2

```
OUTRES,ERASE
OUTRES,ALL,NONE
OUTRES,NSOL,xxx
OUTRES,RSOL,xxx
OUTRES,NLOAD,xxx
OUTRES,STRS,xxx
OUTRES,EPEL,xxx
```

2019 R1

```
OUTRES,ERASE
OUTRES,ALL,NONE
OUTRES,NSOL,xxx
OUTRES,RSOL,xxx
OUTRES,NLOAD,xxx
OUTRES,STRS,xxx
OUTRES,EPEL,xxx
OUTRES,ETMP,xxx
OUTRES,CONT,xxx
OUTRES,NLDAT,xxx
```

Extra OUTRES commands added

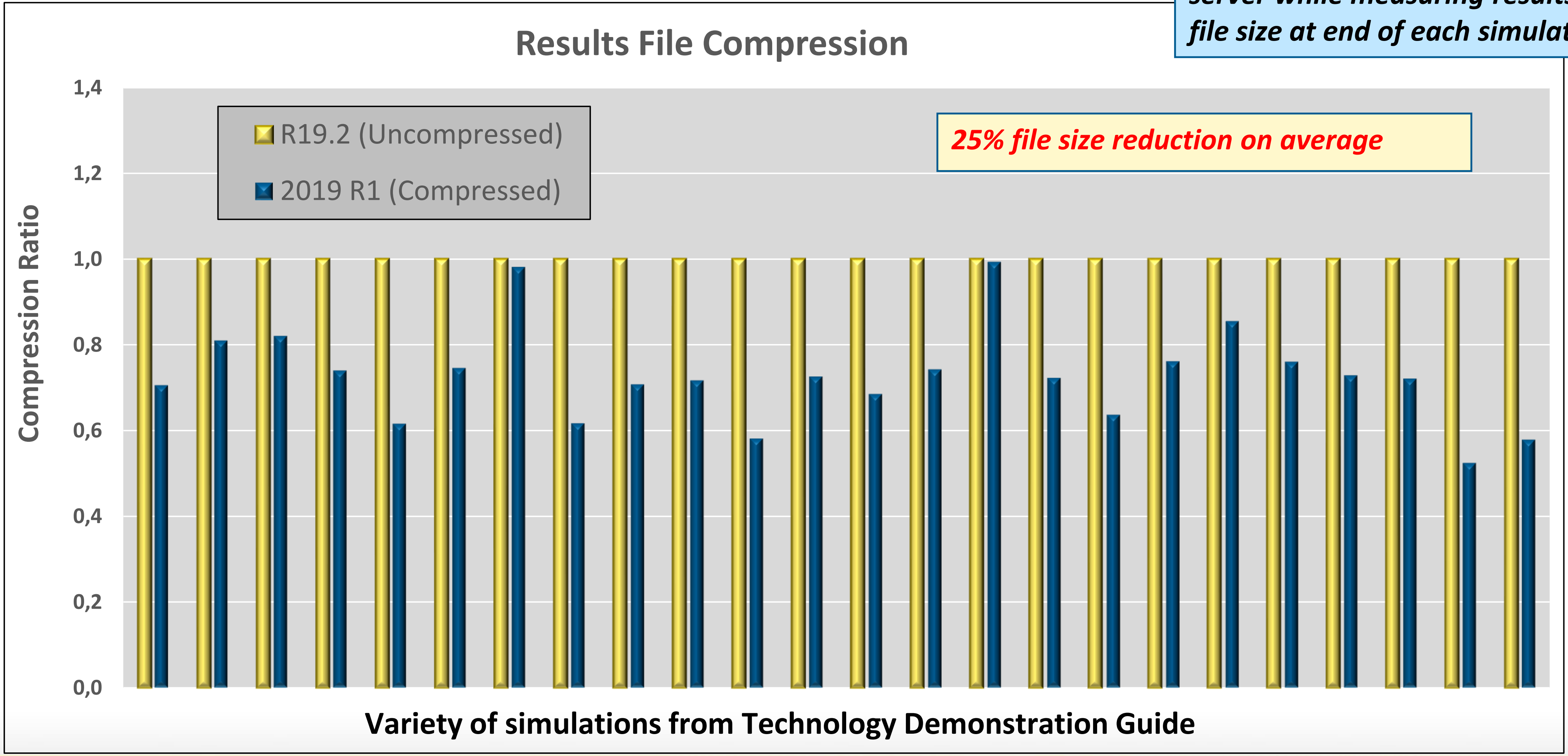
When used in 2019 R1, temperatures, contact data, etc.. are no longer written

Advocate caution when reusing .dat, .inp files from earlier version in 2019 R1

Results File Enhancements

Results file compression (sparsify)

Technology Demonstration Manual models run on Linux server while measuring results file size at end of each simulation



Miscellaneous Enhancements

- **AVL/Excite interface writes out damping matrix (if applicable) to .exb file**
- **MSC/Adams interface for .mnf file has been upgraded**
  - Code has been broken since R19.1 → fixed in this release
  - Some additional items are now written to .mnf file

Mechanical APDL
2019 R1

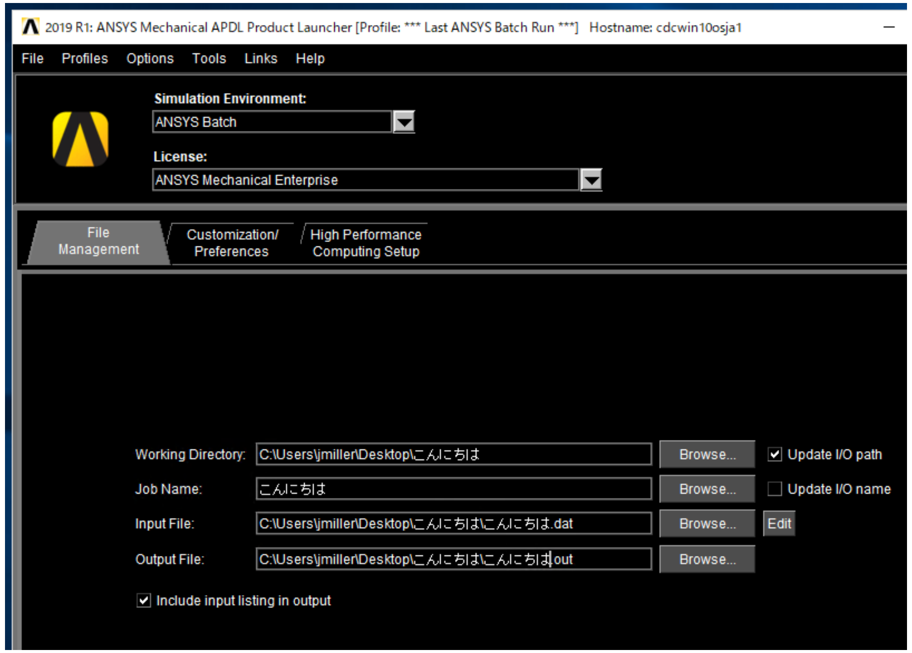
Usability Enhancements

- **Support added for Japanese/Chinese languages**
  - Input file can have Japanese/Chinese characters
    - Ex. –i こんにちは.dat
  - Jobname can be defined with Japanese/Chinese characters
    - Ex. ex. –j こんにちは
  - Directory paths containing Japanese/Chinese characters
    - Ex. –dir C:\Users\user\Desktop\こんにちは\
  - Output file can have Japanese/Chinese characters
    - Ex. –o こんにちは.out
  - *NOTE: must continue to use ASCII characters (i.e., English) for file extensions*

Usability Enhancements

- **Support added for Japanese/Chinese languages**
  - Support added for command line, launcher and certain MAPDL commands

<u>MAPDL commands:</u>
/input,こんにちは.dat
/filname,ファイル
/assign,rst,結果



<u>Command Line Use:</u>
ansys193 –b –dir .\こんにちは –i こんにちは.dat –o こんにちは.out –j こんにちは

Contact and FMDY

ANSYS 2019 R1 update

Distributed (DMP) Contact

Goal:

To improve DMP scalability under higher core counts for contact model with large contact pairs (large number of contact elements relative to the total number of elements in the entire model)

Basic Idea:

It automatically decomposes the large contact pair into sub-pairs, transfers the sub-pairs to different cores, which improves load balance among all CPU cores.

Challenge:

Making identical results between no-splitting and splitting

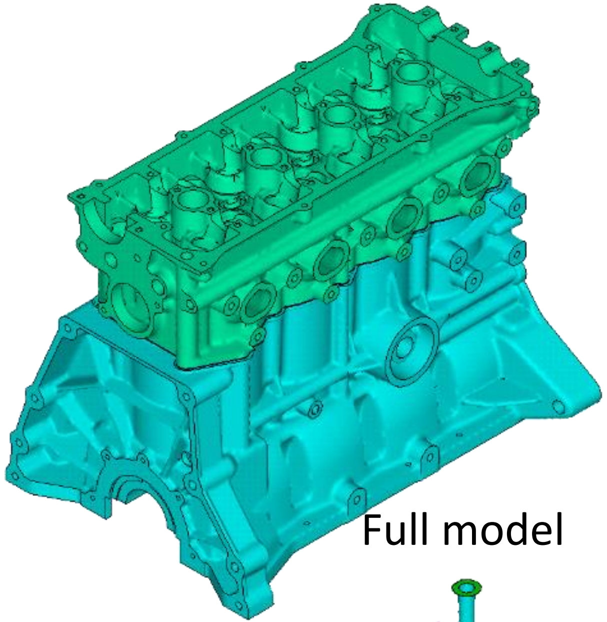
WB/Mechanical: can just post contact results on the original contact pairs without being noticed any split pairs.

New Options in CNCHECK Command

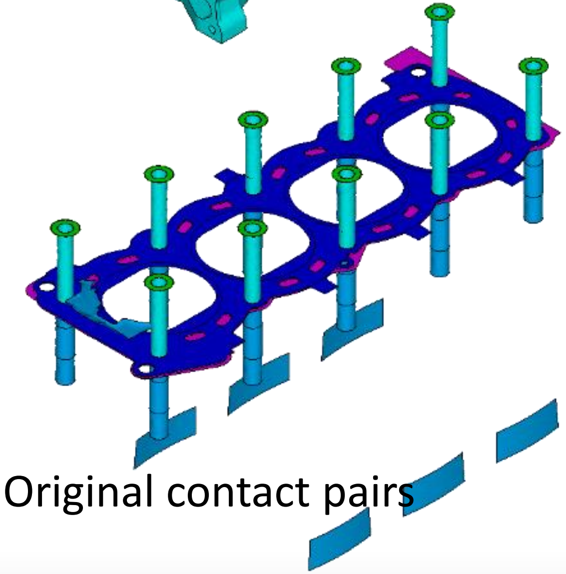
CNCHECK, Option, RID1, RID2, RINC, InterType, Trlevel

SPLIT	Split any original (or base) contact pair into smaller sub-pairs in /PREP7 phase which is mainly for better scalability in DMP run. The split contact pairs may create additional overlapped contact elements at split boundaries. Contact pairs can only be split once. A repeated use of this option has no further splitting for the already split contact pairs.
DMP	This option is similar to the SPLIT, but it is more automatic and contact pair splitting is done at the solution level (**SOLVE**) of the first load step, not at the preprocessing level. This option is valid only in a distributed-memory parallel (DMP) run. For this option, *TRlevel* and *InterType* are valid; all other arguments are ignored.
MERGE	Merge all sub-contact pairs which are previously split by prior *Option* = SPLIT, DMP back to their original pairs. If this option is used, other labels will not work.

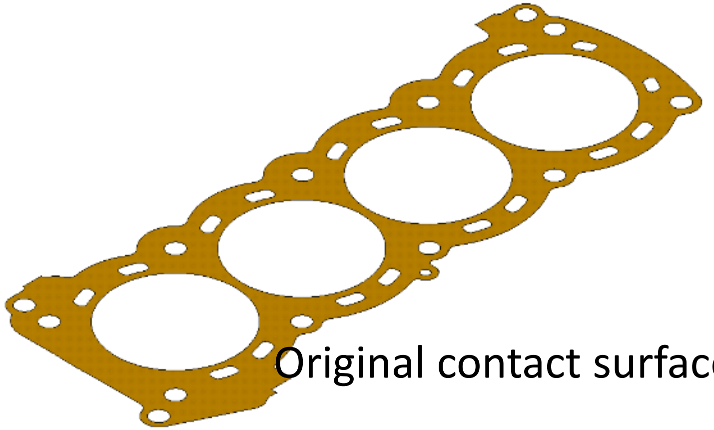
Engine Block with Gasket



Full model



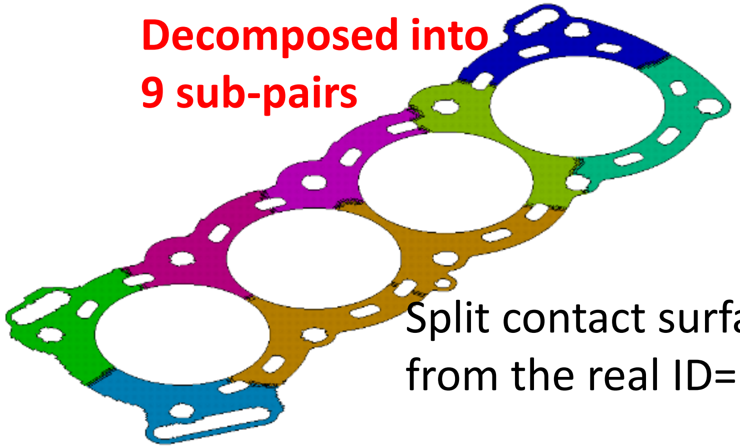
Original contact pairs



Original contact surface

Original target surface

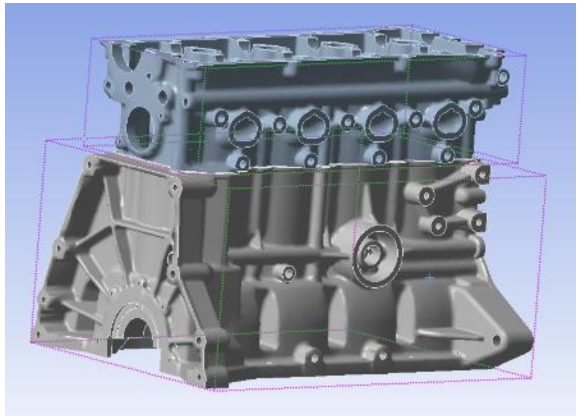
Decomposed into 9 sub-pairs

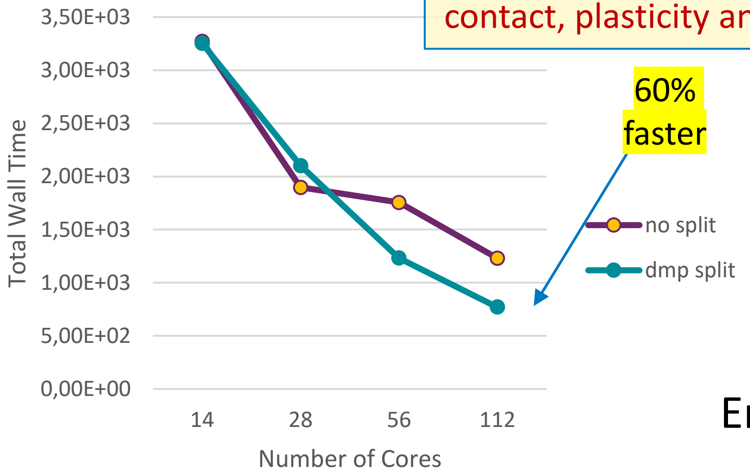


Split contact surfaces from the real ID=18

Trimmed target surfaces from the real ID=18

Examples:



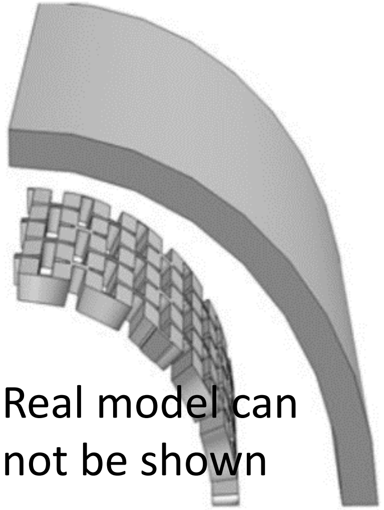


- 3.8 million DOF; sparse solver
- Nonlinear static analysis involving contact, plasticity and gasket elements

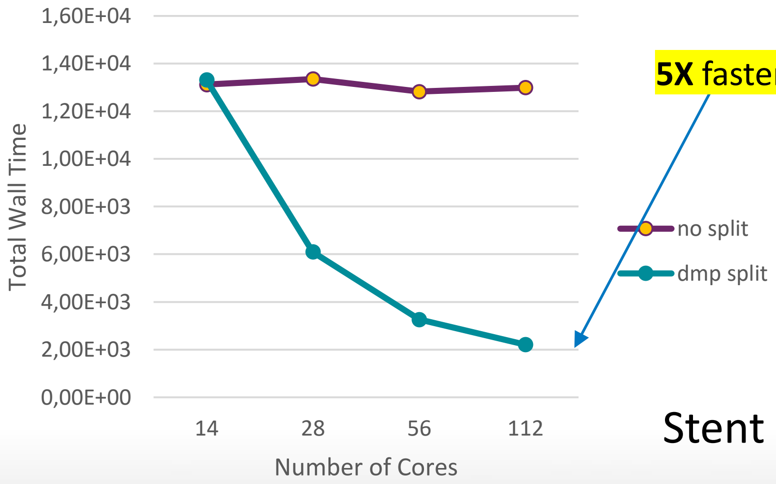
60% faster

Performance gain expected at higher core counts

Engine Block with gasket



Real model can not be shown



5X faster With CNCH,TRIM,,,,SELF

Performance gain expected at higher core counts

Stent Deployment

DMP Contact: Other Customer's Models

Run 02 set with
Number of splitting = 4
Number of CPU cores= 16

Performance gain expected at higher core counts

Models	No-Split	With Split	Time reduction
model-1	6967	5363	30
model-2	26756	14584	83
model-3	37756	22204	Semiimplicit 60
model-4	8463	7414	14
model-5	17439	12559	39
model-6	3259	1871	74
model-7	5664	2685	111
Model-8	155	101	Semiimplicit 53
model-9	4933	1372	260
model-10	1833	1519	21

Semi-Implicit Method To Improve Robustness of MAPDL

Goal:

- To solve complex problems that encounter convergence difficulty in an implicit solver by switching to explicit time integration
- Support most features (e.g. CEs, LMs, Higher order elements, Joints, UP) of MAPDL

Basic Idea:

Central difference time integration (explicit/semi-implicit) transforms governing equations in an explicit form that does not need Newton-Raphson iterations

Semi-Implicit Method guarantees a solution

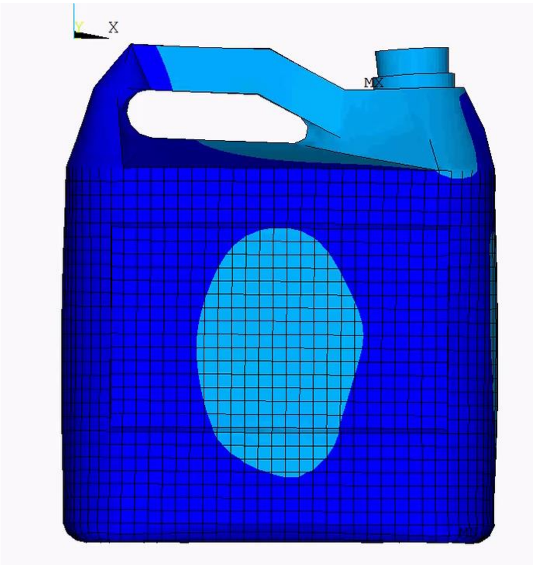
- Start with Implicit
- Convergence difficulty ?-> Change to explicit to overcome hump
- Past the most non-linear part?-> Change back to implicit solver

Command for Semi-implicit Method

Semiimplicit, Option, Type, Value

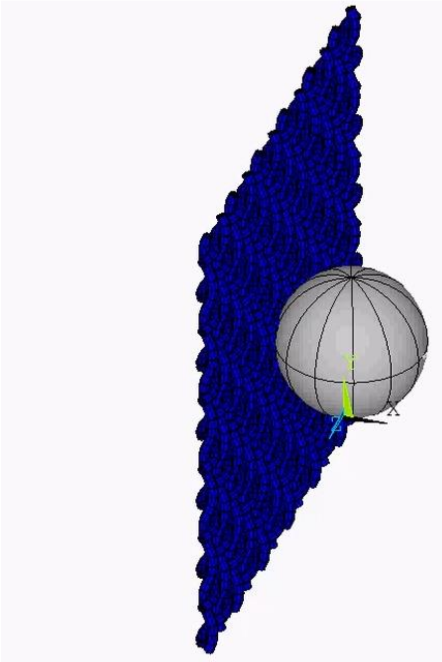
MSCA	**Selective Mass Scaling Factor for the explicit solve phase.** **Selective Mass scaling is needed to run quasi-static problem in explicit** **Type = DTIM, Value = Desired initial minimum time increment** **Type = MASS, Value = Value of Selective mass scaling Factor**
ETOI	**Sets criterion for transitioning from explicit to implicit** **Type= Time,** Value= Time to be spent in explicit
SFAC	**Safety factor for time incrementation in the explicit solve phase**

Semi-Implicit Method To Improve Robustness of MAPDL



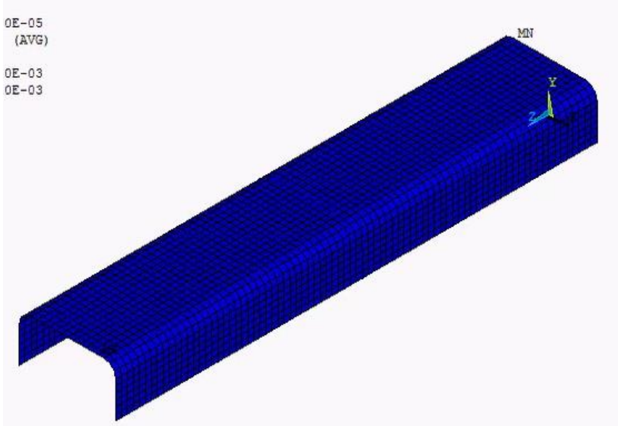
Buckling of a Bottle

Cannot solve the problem in implicit alone- has convergence difficulties
Automatically transitions to explicit and solves the problem in explicit



Tennis Ball impacting a net

Automatically transitions from implicit to explicit and back to implicit 9 times



Crushing of Thin Shell Box

Automatically transitions from HHT to explicit (half way) and solves the problem in explicit.

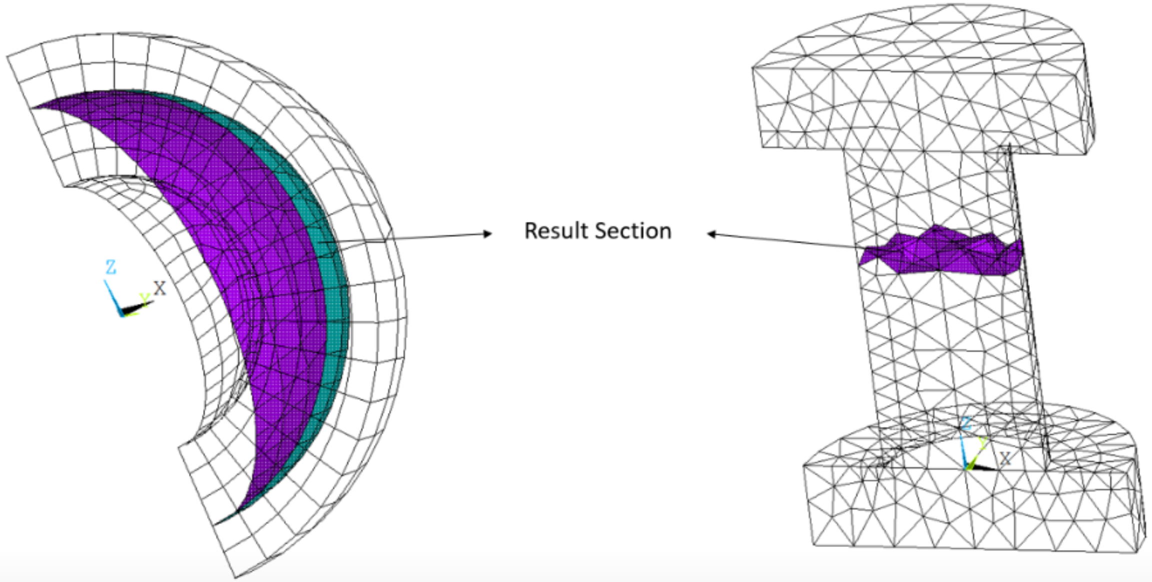
ANSYS

Result Section Output

A result section is a user-defined surface that can be used to output and monitor section forces, moments, heat flow, current, mass flow, and pore fluid volume flux during a solution.

Result section output quantities (like bending moments and axial forces etc.) can be monitored without storing results for all elements for each sub step, which significantly reduces the results file size.

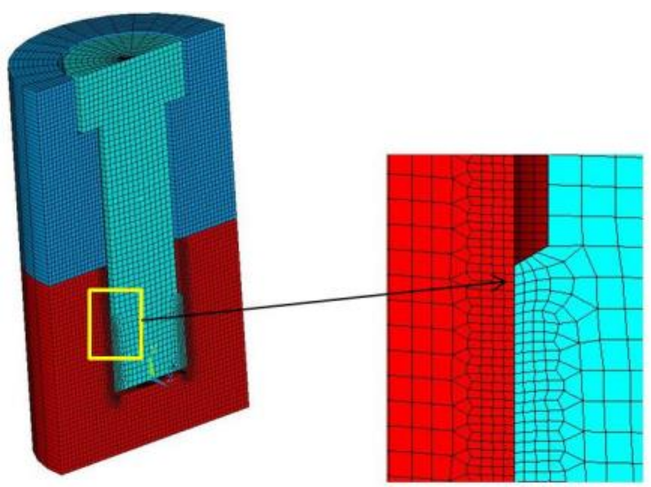
RSMESH	automatically creates the result section inside 2-D and 3-D continuum element meshes.
OUTPR,RSFO	The section output quantities are written to a single `Jobname.SECF` text file at a user-specified frequency
NLHIST	Certain result section quantities can also be monitored (or terminated) during solution

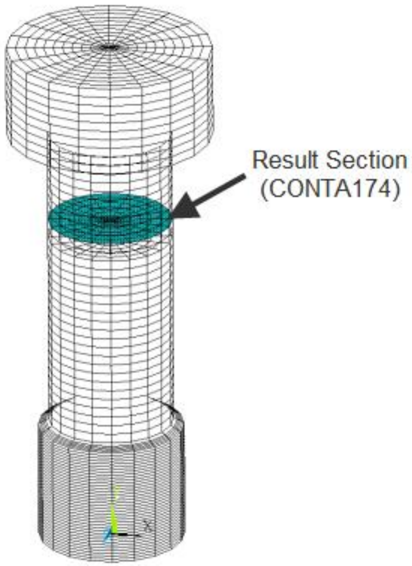


Example: Result Section in a Bolt Thread Model

This 3-D model represents an M120 structural steel bolt with standard thread dimensions.

The model includes a cover plate and a base plate. A bilinear isotropic plastic material model is used for the bolt and the plates.





3-D Half Model of a Simplified Bolt Joint

```
/PREP7
ESEL,S,TYPE,,105     ! Select the bolt elements
RSMESH, , ,,y,280,,,elgr
/SOLU
OUTPR,RSFO,5        ! Define the frequency to output results
```


```
<SOLUTION>
<HEADER FRQ="SUBSTEP">
 <COLUMN ID="     1">Result Section ID</COLUMN>
 <COLUMN ID="     2">Total Section Force</COLUMN>
 <COLUMN ID="     3">Normal Section Force-x</COLUMN>
 <COLUMN ID="     4">Tangential Section Force-y</COLUMN>
 <COLUMN ID="     5">Tangential Section Force-z</COLUMN>
 <COLUMN ID="     6">Total Section Moment</COLUMN>
 <COLUMN ID="     7">Normal Section Moment-x</COLUMN>
 <COLUMN ID="     8">Tangential Section Moment-y</COLUMN>
 <COLUMN ID="     9">Tangential Section Moment-z</COLUMN>
 <COLUMN ID="    10">Deformed Section Area</COLUMN>
 <COLUMN ID="    11">Deformed Section Diameter</COLUMN>
 <COLUMN ID="    12">Section Axial Stress</COLUMN>
 <COLUMN ID="    13">Section Bending Stress</COLUMN>
 <COLUMN ID="    14">Section Center X-Coor</COLUMN>
 <COLUMN ID="    15">Section Center Y-Coor</COLUMN>
 <COLUMN ID="    16">Section Center Z-Coor</COLUMN>
 <COLUMN ID="    17">Section Normal-X</COLUMN>
 <COLUMN ID="    18">Section Normal-Y</COLUMN>
 <COLUMN ID="    19">Section Normal-Z</COLUMN>
 <COLUMN ID="    20">1st rotation about local Z</COLUMN>
 <COLUMN ID="    21">2nd rotation about local X</COLUMN>
 <COLUMN ID="    22">3rd rotation about local Y</COLUMN>
 <COLUMN ID="    23">Heat Flow</COLUMN>
 <COLUMN ID="    24">Current Flow</COLUMN>
 <COLUMN ID="    25">Diffusion Flow Rate</COLUMN>
 <COLUMN ID="    26">Fluid Flow</COLUMN>
 <UNITS>UNDEF</UNITS>
</HEADER>
<COLDATA LOAD_STEP="       1" SUBSTEP="       5" ITERATION=
      103     580511.4        -580507.4        -1234.491
</COLDATA>
<COLDATA LOAD STEP="       1" SUBSTEP="      10" ITERATION=
```


Result Section in the Bolt at y = 280

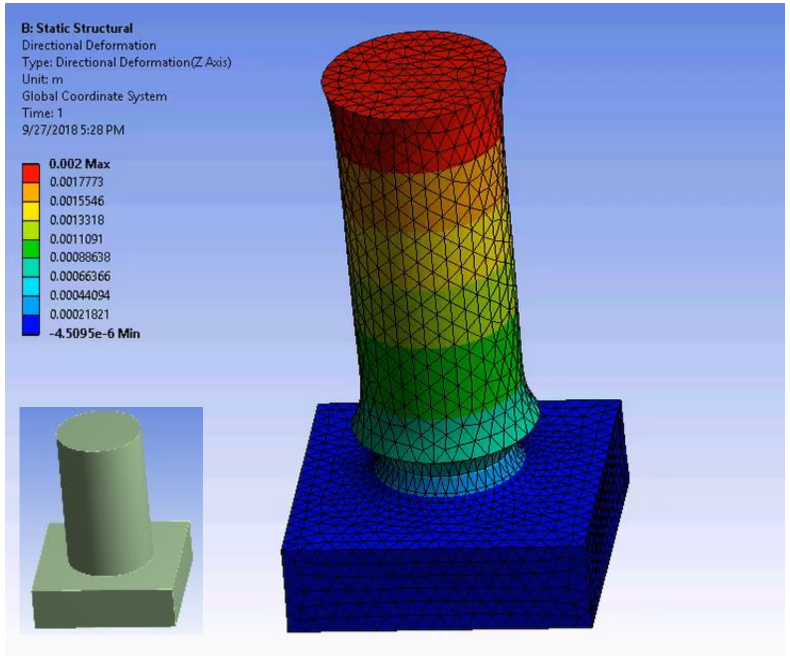
SMART Crack Growth

ANSYS 2019 R1 update

SMART Crack Growth Simulation Framework

New SMART crack growth enhancement

- **Support temperature loading**
    - Map temperature boundary condition
- **Support surface pressure loading**
    - Map pressure load for external surfaces and current crack surfaces
    - Allow to define pressure load for newly generated crack surfaces
- **Support tabular pressure load as function of time**
- **Support both SMP and DMP**
- **Support PCG and SPARSE solvers**
- **Support displacement boundary condition mapping**
    - Fixed DIS boundary condition
    - Face based DIS boundary condition



SMART Crack Growth Simulation Framework

New crack surface pressure load

- Command to specify new crack surface pressure
  - CGROW,CSFL,loadtype, loadkey,value
  - loadtype: type of load, currently only pressure load as PRESSURE
  - loadkey: load key for the crack surface load, current set to 0
  - Value: the new crack surface load value for the load step
    - **Current only constant load value can be applied**
    - **To define a varying load, use table function %table%, which table defines pressure load table as function of time (and may be in the future as function of coordinates x, y, z). Use *DIM to define the table.**

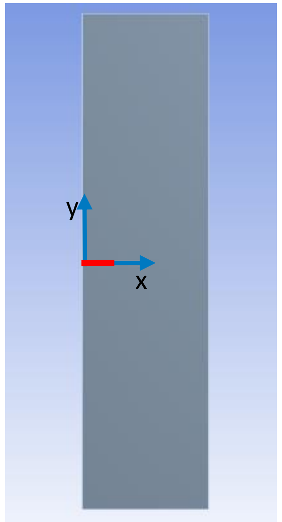
```
prsv=20000000 ! unit Pa (400MPa)
! apply presure to new crack surface
cgrow,csfl,press,,prsv
```

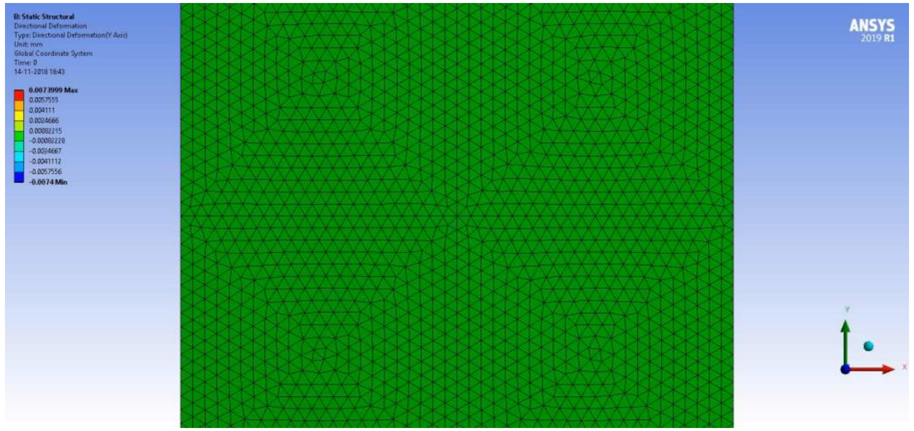
SMART Crack Growth Simulation Framework

Problem description

- Edge crack panel subject to a thermal load
- Static crack growth with J integral as criterion

Linearly varying temperature from left to right,
$T(x)=(2x/W)*T_0$, $T_0=20\ ^oC$,
Reference temperature = 0 ^{o}C

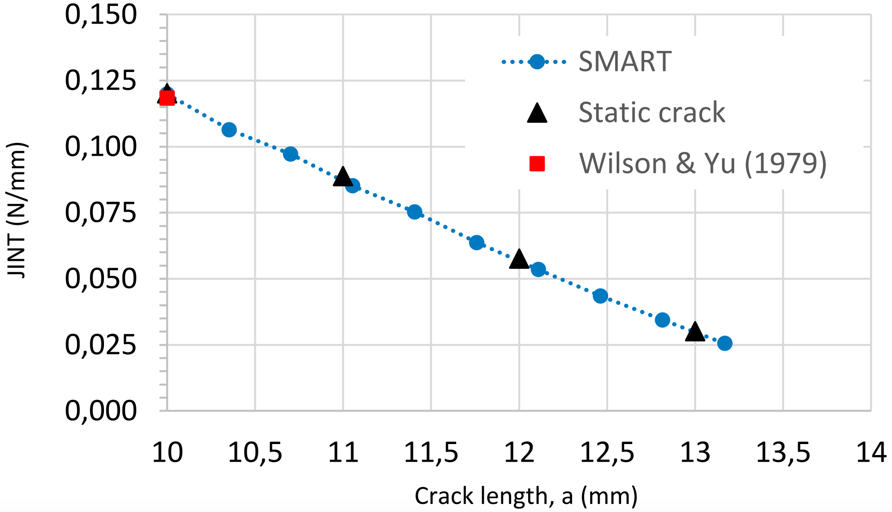


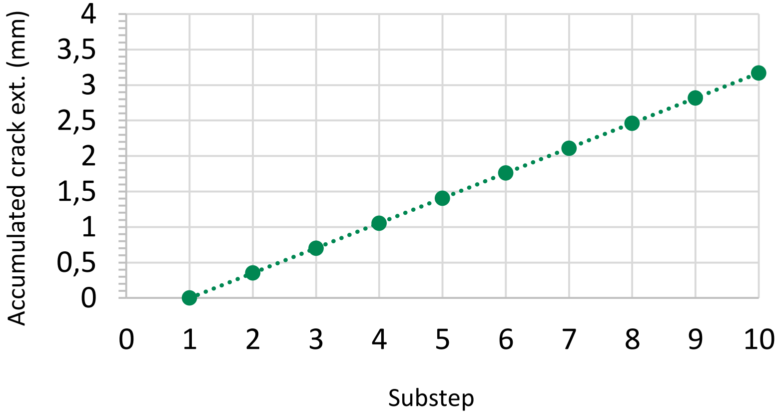


Boundary conditions:

- At top and bottom faces, **Ux=Uy=0**
- At all nodes, **Uz=0** (plane strain)

Plane strain boundary condition is maintained by the mapping of the displacement boundary condition.





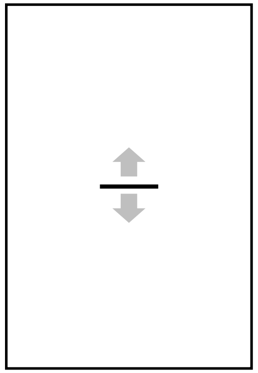
(*Wilson, W.K. and Yu, I.W., 1979. The use of the J-integral in thermal stress crack problems. *International Journal of Fracture*, *15*(4), pp.377-387)

SMART Crack Growth Simulation Framework

Center crack panel subjected to surface pressure

Problem description

- Tensile panel with a center crack
- Static crack growth with J integral as criterion



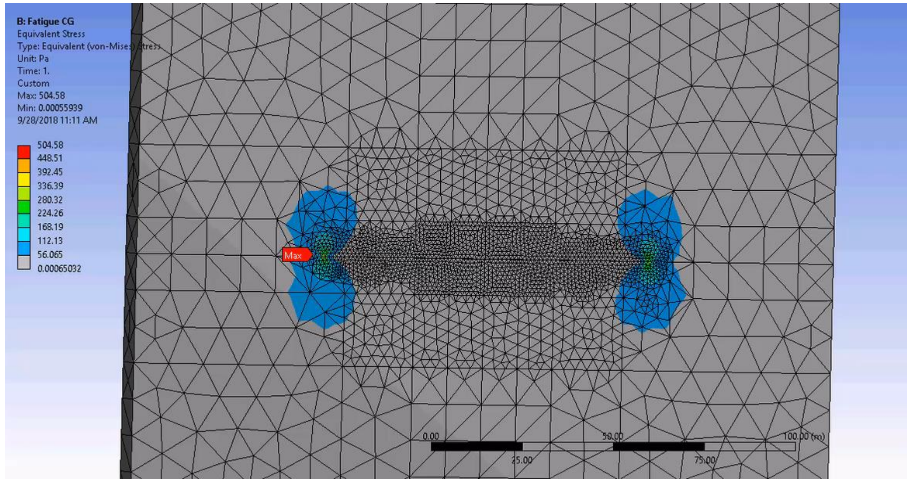
$$K_I = p\sqrt{\pi a}\, F\left(\frac{a}{W}\right)$$

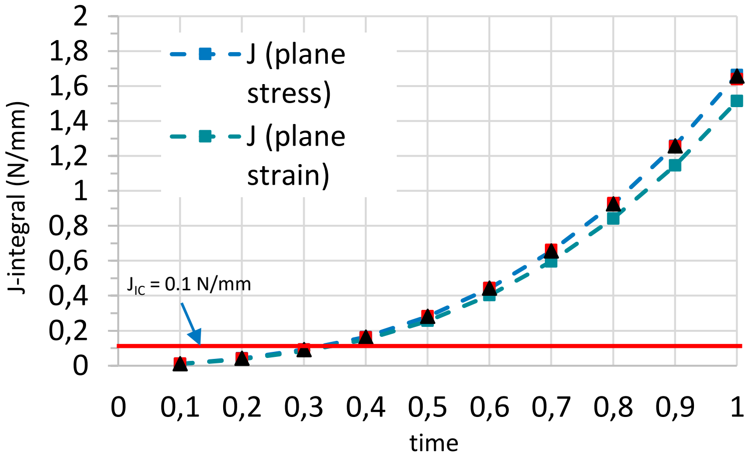
$$F\left(\frac{a}{W}\right) = \frac{1 - 0.5\left(\frac{a}{W}\right) + 0.370\left(\frac{a}{W}\right)^2 - 0.044\left(\frac{a}{W}\right)^3}{\sqrt{1 - a/W}}$$

(The Stress Analysis of Cracks Handbook, ASME, 2000, Page 41)

$$J = \frac{K_I^2}{E} \qquad \text{(plane stress)}$$

$$J = (1 - \nu^2)\frac{K_I^2}{E} \qquad \text{(plane strain)}$$



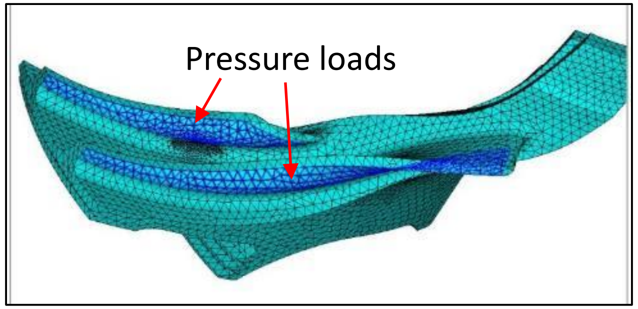


J (average of 5 contours) evaluated at the middle of the crack front

SMART Crack Growth Simulation Framework

Distributed Solution

- **Work within ANSYS distributed solution architecture**
- **FEM model solved in distributed mode**
- **Fracture and SMART Crack Growth Calculation including remeshing are conducted only in master node**



Solution 1

Elements: 0.058M
Equations: 0.23M
Crack tips : 23
Remeshing: 5 times

# of CPU	Time (seconds)	Speedup
1	163	1
2	109	1.50
4	82	1.98
8	70	2.32
16	72	2.26

Solution 2

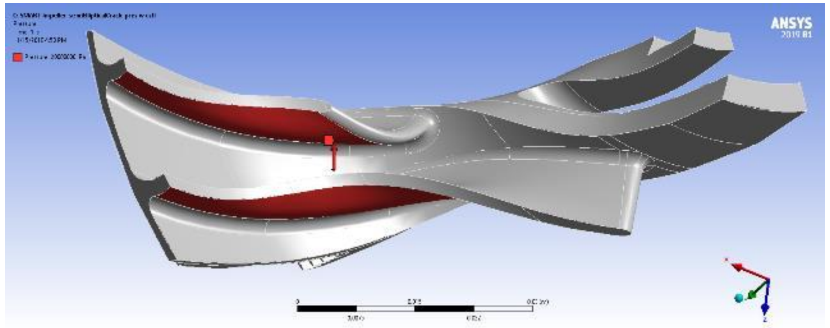
Elements: 0.67M
Equations: 2.7M
Crack tips : 101
Remeshing: 3 times

# of CPU	Time (s)	Speedup in total
1	8799	1
8	2010	4.38
16	1327	6.63
32	991	8.88

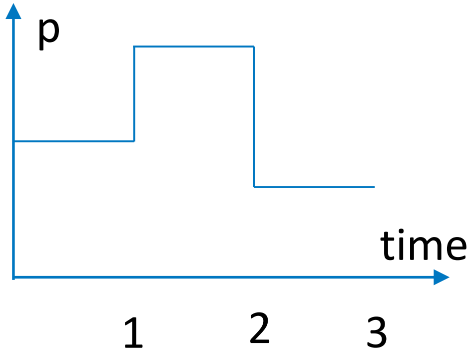
SMART Crack Growth Simulation Framework

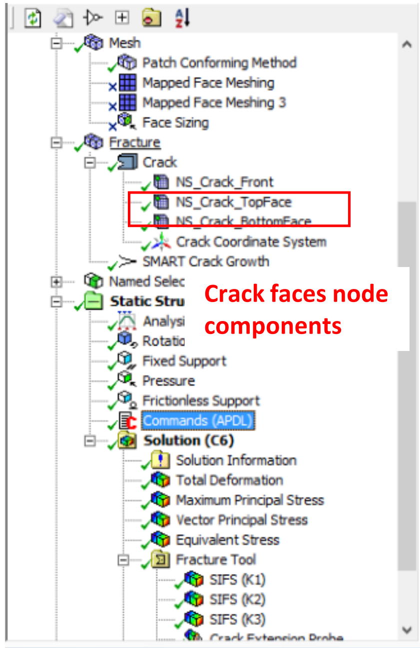
Multi load step and tabular load

- Various load magnitude for the fatigue



```
*dim,prsload,table,6,1,,TIME
prsload(1,0) = 0
prsload(1,1) = 100
prsload(2,0) = 1.0
prsload(2,1) = 100
prsload(3,0) = 1.0001
prsload(3,1) = 160
prsload(4,0) = 2
prsload(4,1) = 160
…
```






```
prsv=20000000              ! unit Pa (400MPa)

! Apply pressure to crack surface
cmsel,s,NS_Crack_TopFace,node
cmsel,a,NS_Crack_BottomFace,node
SF,all,pres,prsv

! apply surface pressure to new crack surface
cgrow,csfl,press,,prsv
```

```
! Apply tabular pressure to crack surface
cmsel,s,NS_Crack_TopFace,node
cmsel,a,NS_Crack_BottomFace,node
SF,all,pres,%prsload%

! apply surface pressure to new crack surface
cgrow,csfl,press,, ,%prsload%
```


SMART Crack Growth Simulation Framework

Solution results output control

- Use OUTRES command to control for different results output

```
OUTRES,ALL,30        ! Write solution to rst file for every 30 substeps
OUTRES,CINT,ALL      ! Write CINT (fracture) results to rst file for all substeps
```

SMART crack growth usually generates a large number of substeps, writing results at every substep will result in a very large rst file.

Mechanical Enhancements

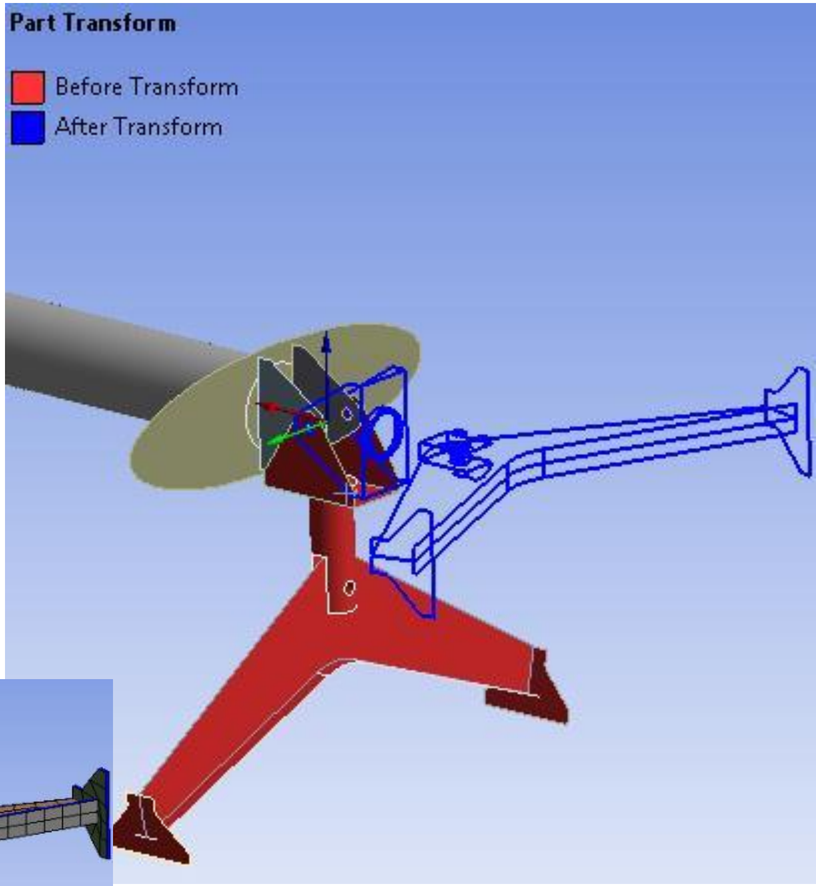
ANSYS 2019R1 update

Part Transform

The new Part Transform feature that allows you to reorient parts by specifying translations and/or rotations within Mechanical

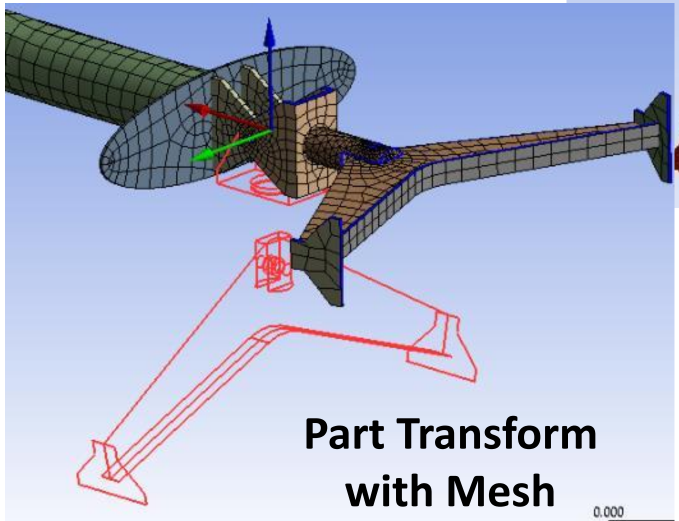
Allows you to transform mesh and/or automatically regenerate contacts along with transforming parts

Graphical preview allows you to see the location of the parts before and after the transform



Details of "Part Transform"

Scope	
Scoping Method	Geometry Selection
Geometry	3 Parts
Definition	
Define By	Rotation And Translation
Coordinate System	Coordinate System
Translate X	0. m
Translate Y	0. m
Translate Z	0. m
Rotate X	0. °
Rotate Y	90. °
Rotate Z	0. °

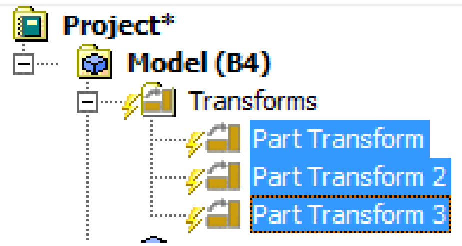


Part Transform with Mesh

Part Transform

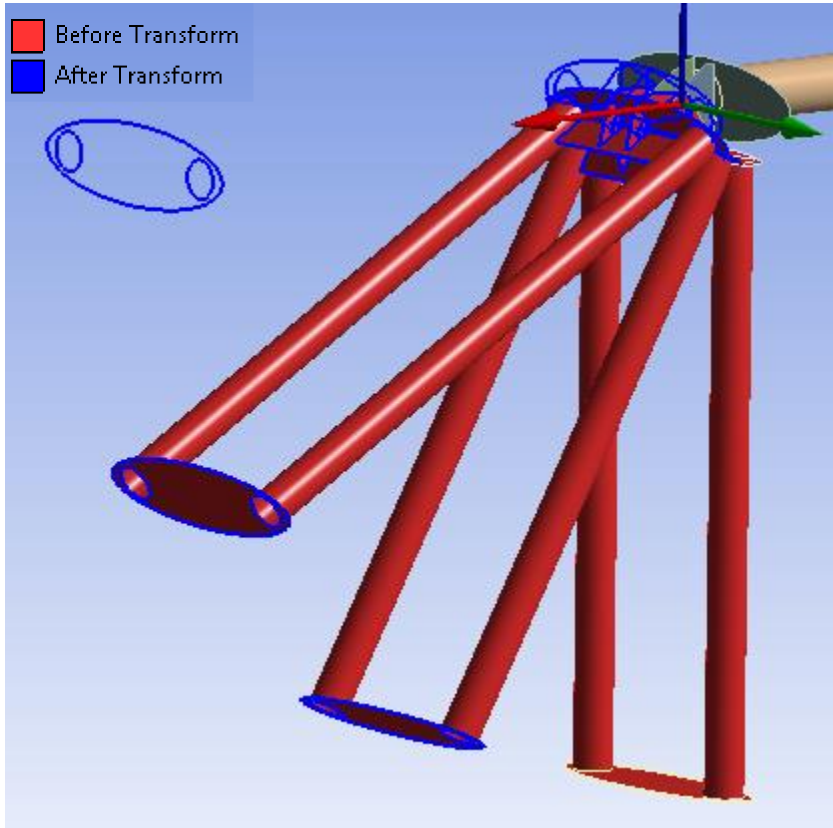
Allows you to reorient one or more parts around arbitrary coordinate systems

Allows you to reorient one or more parts using a pair of coordinate systems (source and target). The application automatically calculates the transform such that the source is aligned with the target after transform.

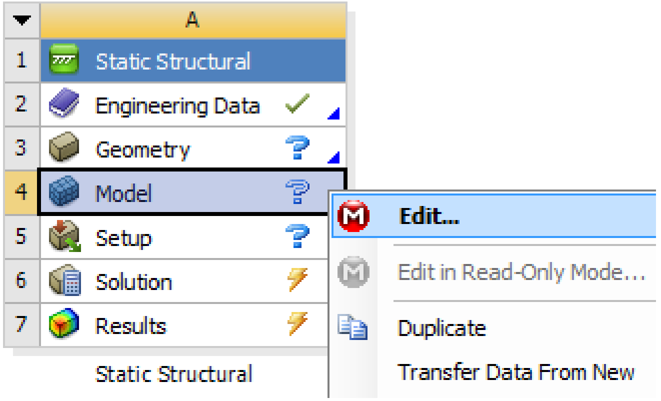


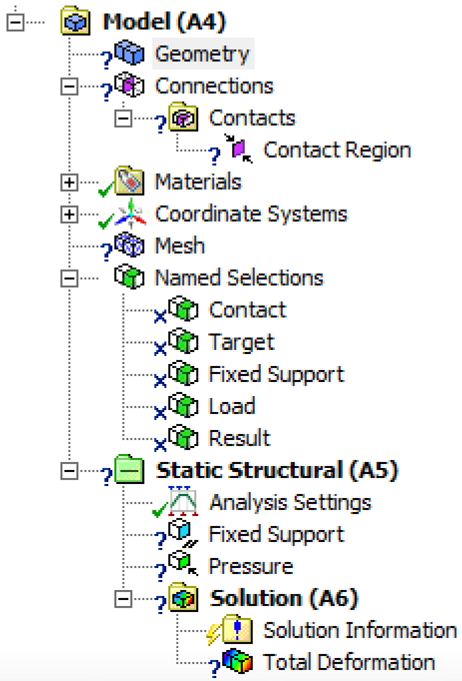
You can apply multiple transforms to a part. The transforms are applied in the order they appear in the tree.

You can reorder the transforms by drag-drop in the tree.



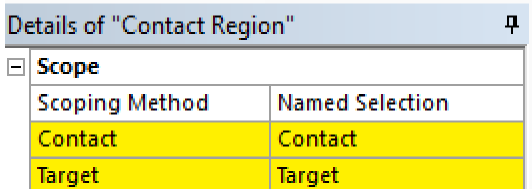
Simulation Template

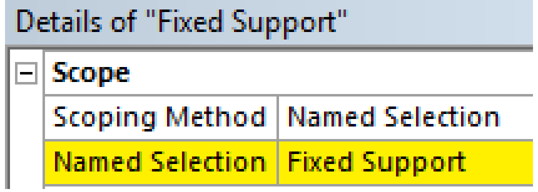


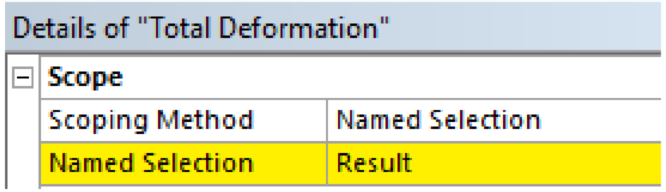


Workbench now allows users to enter/edit Mechanical without attaching geometry.

Users can setup their analysis system using criterion based named selections without having to attach geometry. Such an analysis setup is called a Simulation Template.

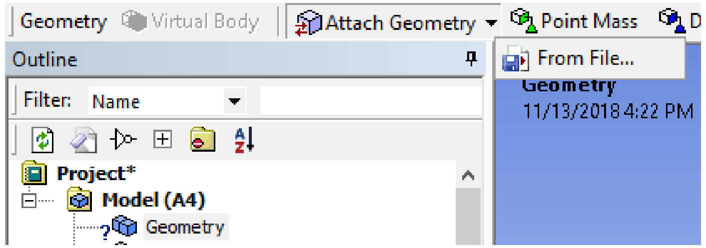


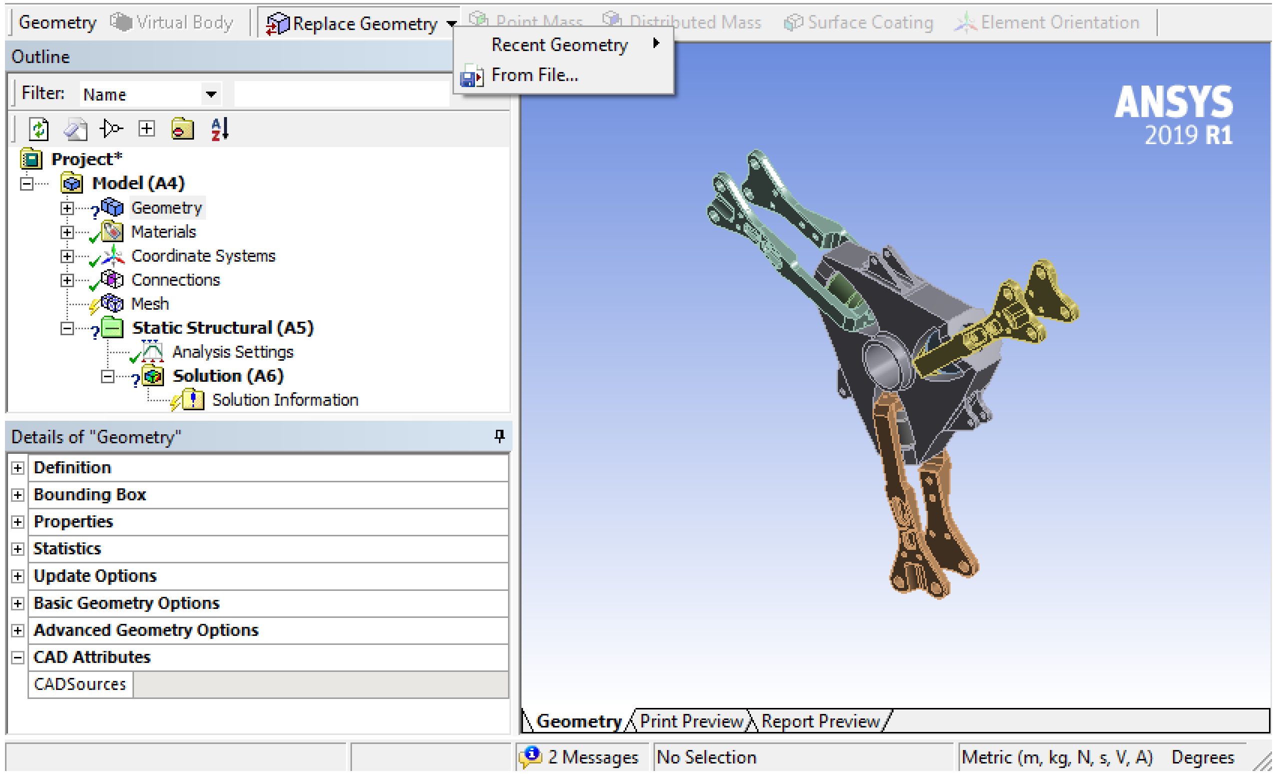




Attach/Replace Geometry

From Mechanical, the new option Attach Geometry, available from the Geometry object toolbar, enables you to import a geometry from with the application.





Once you attach a geometry, or for a system that already includes a geometry, the Replace Geometry option replaces Attach Geometry enabling you to replace an existing geometry.

Convection Fluid Flow

The Convection boundary condition now supports vertex and node scoping when using information from Thermal Fluid line bodies.

This feature enables you to use a specific vertex or node to get the bulk temperature in the convection calculations.

Both direct and coping and named selections are supported

Details of "Convection"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Convection
Film Coefficient	735.91 W/m^2·°C (step applied)
Convection Matrix	Program Controlled
Suppressed	No
Fluid Flow Controls	
Fluid Flow	Yes
Scoping Method	Geometry Selection
Fluid Flow Scoping	1 Node

Details of "Convection"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Convection
Film Coefficient	735.91 W/m^2·°C (step applied)
Convection Matrix	Program Controlled
Suppressed	No
Fluid Flow Controls	
Fluid Flow	Yes
Scoping Method	Named Selection
Fluid Flow Scoping	fluidl

Restart Controls For Nonlinear Adaptivity Analysis

Independent Restart Controls are now available under Nonlinear Adaptivity Remeshing Controls.

The options control the generation/retention of .RDnn remeshing database files, which are needed for mesh nonlinear adaptivity analysis.

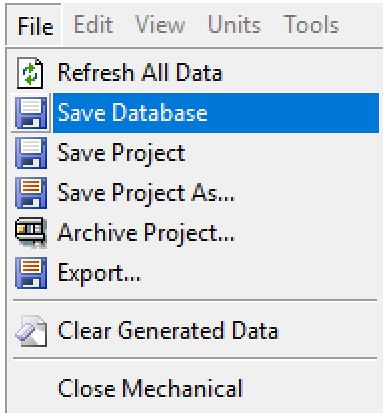
Details of "Analysis Settings"

Step Controls	
Solver Controls	
Rotordynamics Controls	
Restart Controls	
Nonlinear Adaptivity Remeshing Controls	
Refinement Algorithm	General Remeshing
Remeshing Gradient	Practical Shape Gradient
Boundary Angle	15. °
Edge Splitting Angle	10. °
Number of Sculpted Layers	
--Refinement (NSL)	2.
Global Size Ratio	
--Refinement (GSR)	0.75
Remeshing Tolerance	
--Refinement (RT)	0.5
Generate Restart Points	Program Controlled
Retain Files After Full Solve	Yes
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

Miscellaneous Usability Enhancements

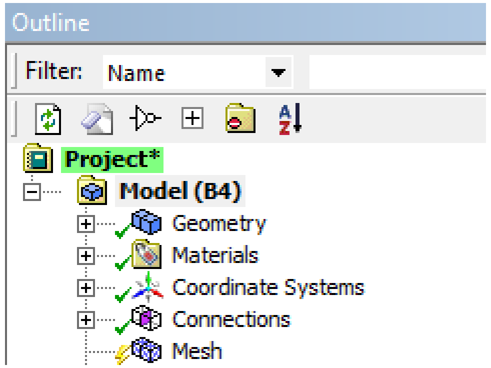
Quickly Save Mechanical Session

- A new File menu option is available: Save Database. This option enables you to save the current Mechanical session without having to save the entire project. However, you must save the project when you exit the application to properly save your changes.



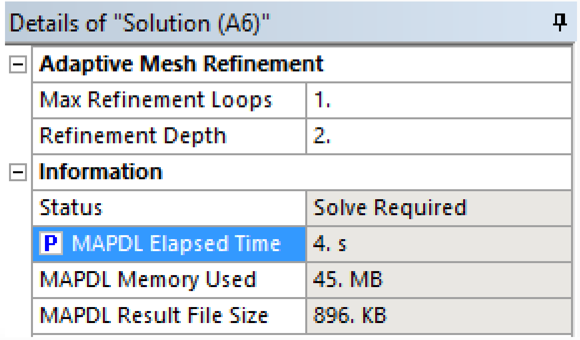
Project Object

- The Project object in the Outline now displays with an asterisk (*) in its name to indicate that you have not yet saved the Mechanical database since the last change or set of changes.



Parametrizing MAPDL Elapsed Time

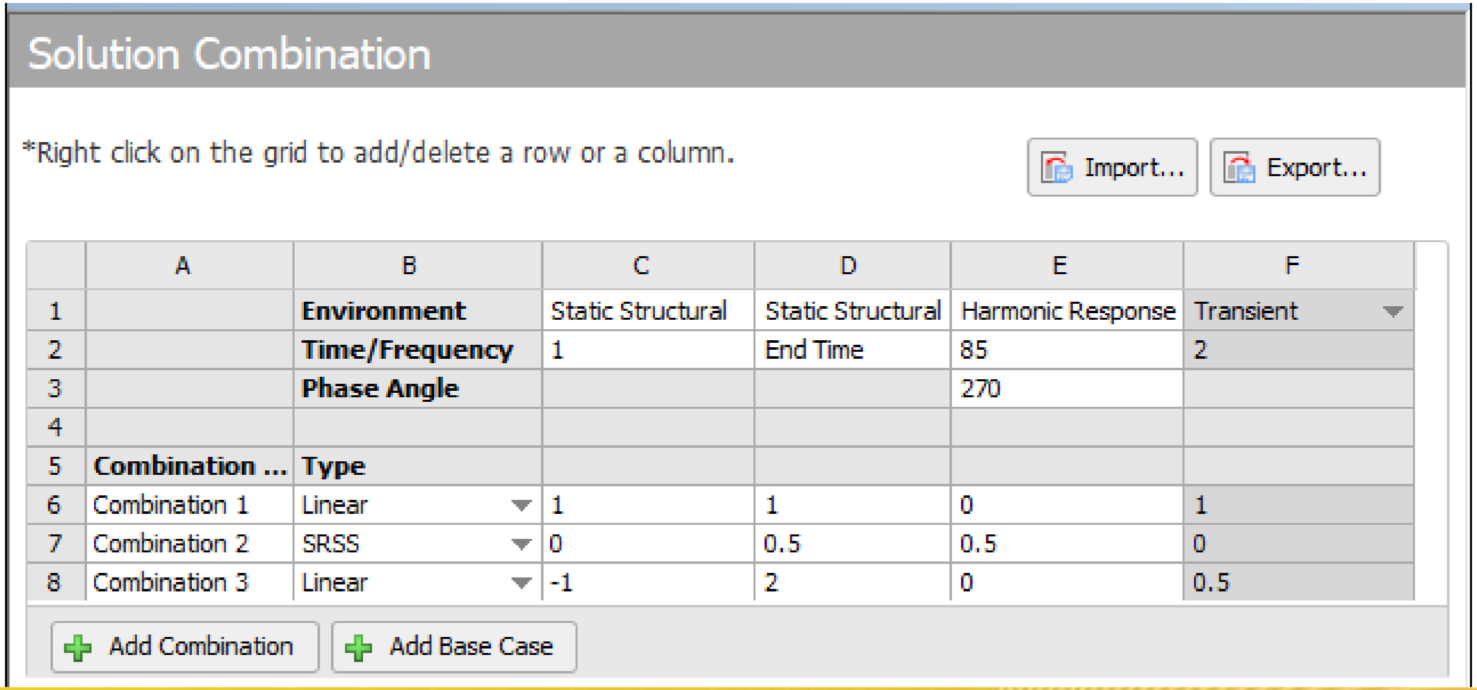
- The MAPDL Elapsed Time can now be parametrized and is available as an output parameter in the Parameter Set in workbench.



New Solution Combination

With the new solution combination, users can:

- Specify **multiple** combinations.
- Combine solutions for **Static** Structural, **Transient** Structural, and **Harmonic** Response analyses.
- Specify solution combinations as either **Linear** or **SRSS** (Square Root of Summation of Squares).
- Use Tabular Data or a result Set Number to specify which combination you wish to display.
- **Import** and/or **Export** the Solution Combination Worksheet as a Comma Separated Value (CSV) file.

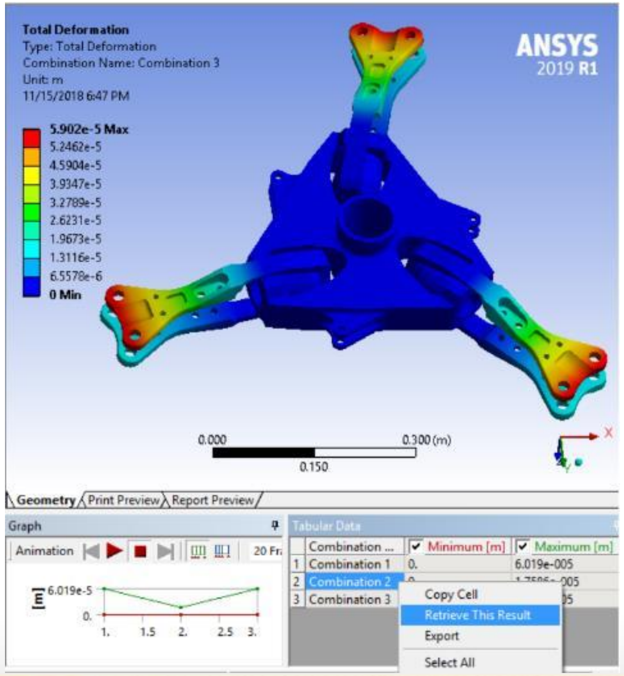

Solution Combination

*Right click on the grid to add/delete a row or a column.

Import... Export...

	A	B	C	D	E	F
1		Environment	Static Structural	Static Structural	Harmonic Response	Transient
2		Time/Frequency	1	End Time	85	2
3		Phase Angle			270	
4						
5	Combination ...	Type				
6	Combination 1	Linear	1	1	0	1
7	Combination 2	SRSS	0	0.5	0.5	0
8	Combination 3	Linear	-1	2	0	0.5

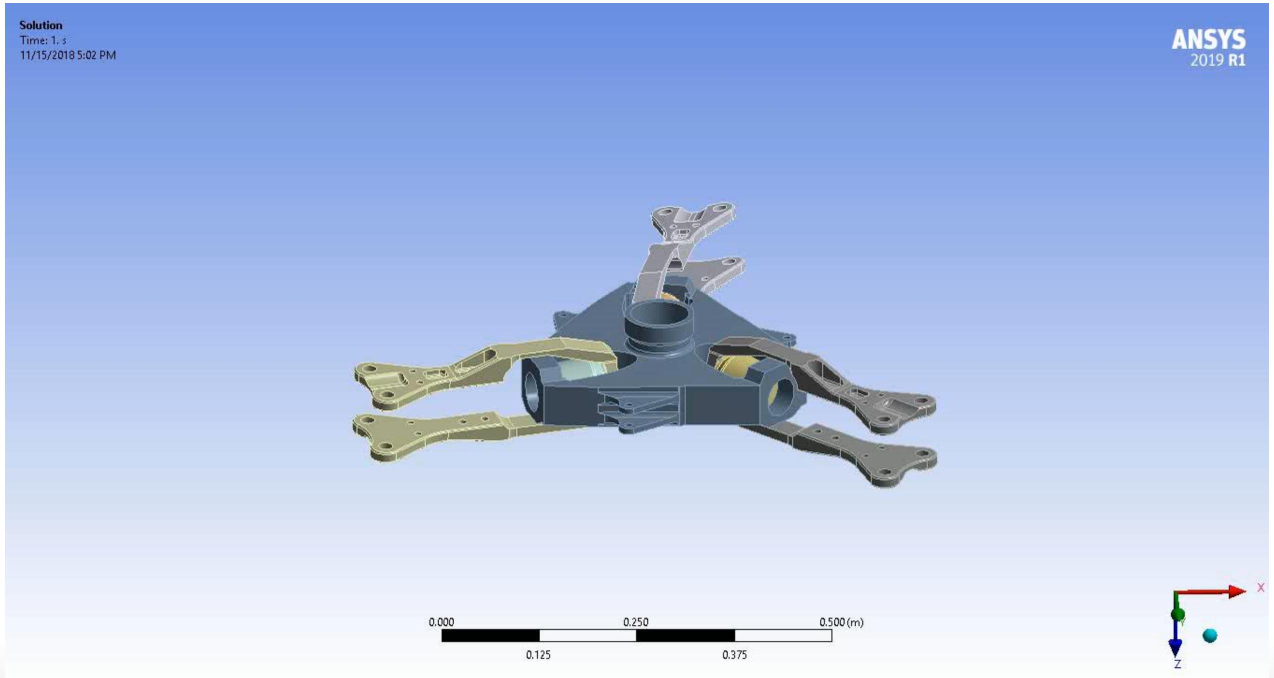
Add Combination Add Base Case

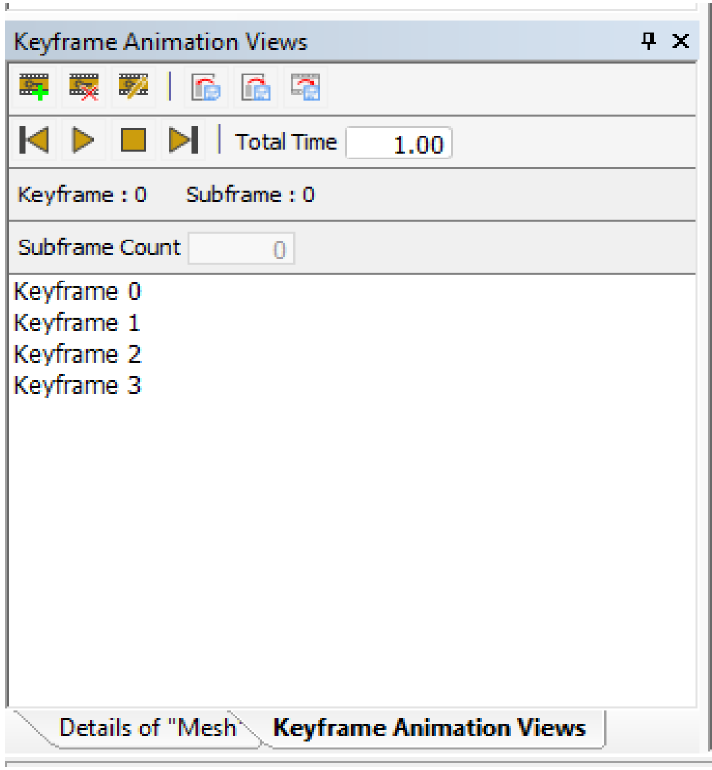


Keyframe Animation

Keyframe animation enables you to string together different snapshots of the model in the Geometry window to create an animation

- Keyframes are created by positioning the model in the desired orientation and clicking on Create Keyframe.
- The application interpolates the transition from keyframe to keyframe to create a smooth animation.

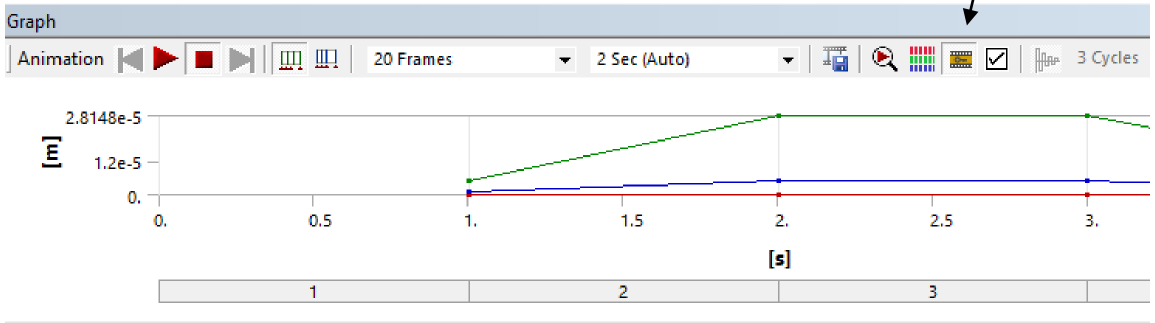


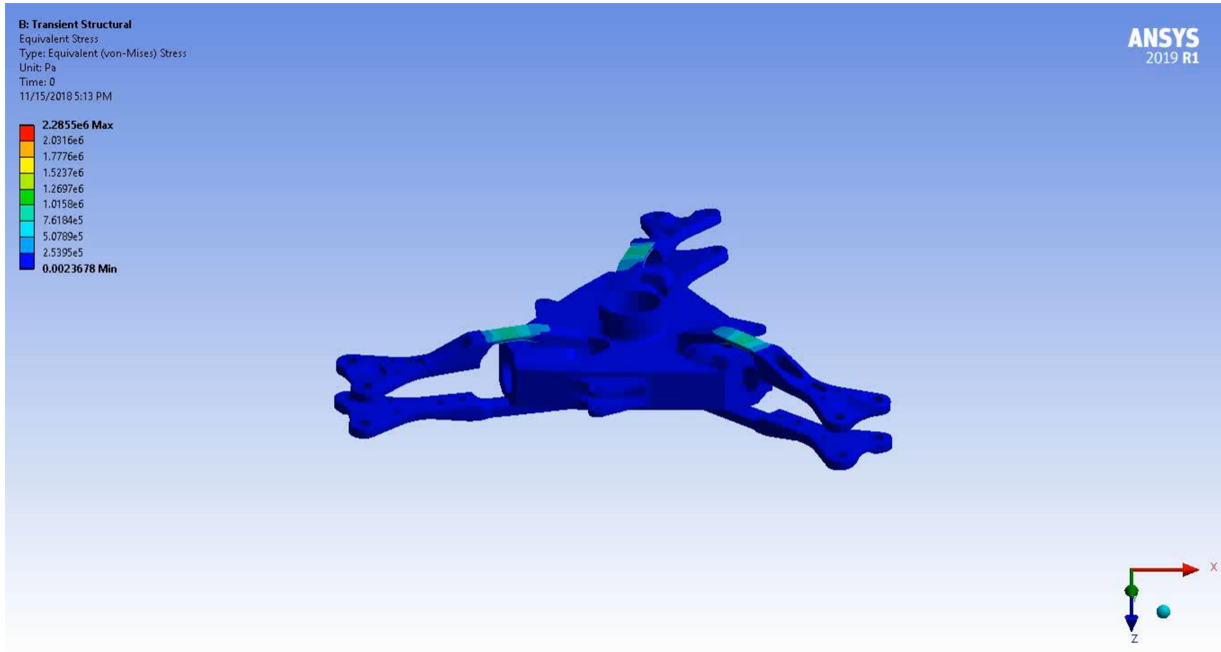


- Export the video in various formats: MP4, WMV , AVI and GIF

Keyframe Animation (continued) for Results

- Keyframe animation for Results enables stitching different snapshots (different view settings) of the model for smooth visualization of results, coordinating the animation in time/steps space (natural for results) with the one in the 3D space (natural for keyframe animation).
- This can be enabled by clicking the keyframe animation icon in the animation toolbar and the user can have a choice of synchronizing the frame rate similar to the one set in the keyframe animation window





Frames Synchronized with Keyframe animation

Export Animation

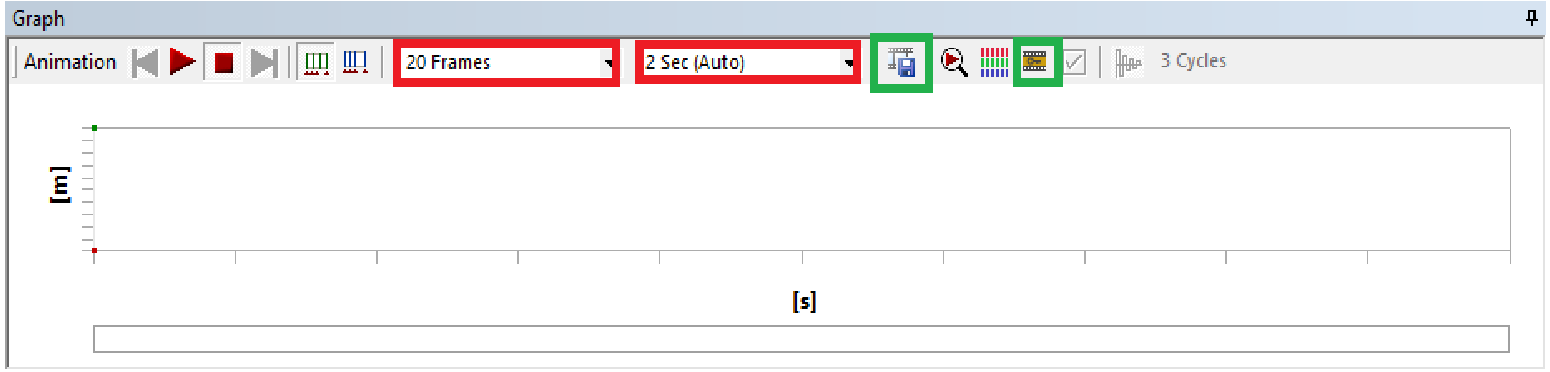
Animation Tool animates the results to show the evolution of the specified result of a simulation.

Keyframe Animation Tool, allows the examination of the models(geometry and result) from different points of view in a dynamic way.

The animation can be exported by selecting the export animation option on the animation toolbar after selecting the result on the solution tree.

The supported formats to export animation are MP4, WMV , AVI and GIF.

- Varying the frame rate allows the capture of a specific number of snapshots to export the video file
- Varying the play length allows the generation of animation video of a specific duration.
- When the keyframe animation for result is enabled the keyframe animation for the solution result is exported as media file

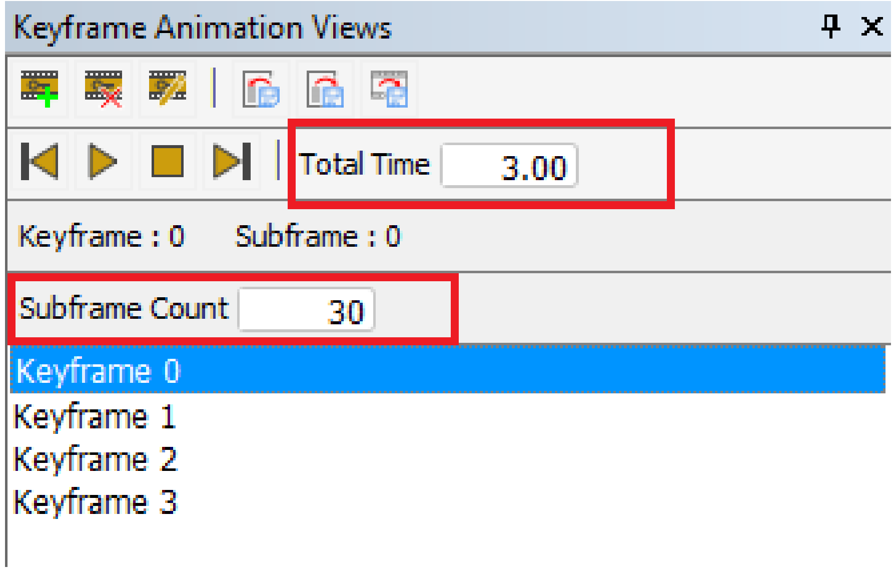


Export Animation (continued)

The generated keyframes can be exported by the export keyframe option on the keyframe animation toolbar.

•The total number of intra frames between main keyframes for exported animation can be specified by the **Subframe Count** .

•The total play duration of exported animation file can be specified by the **Total Time** option .

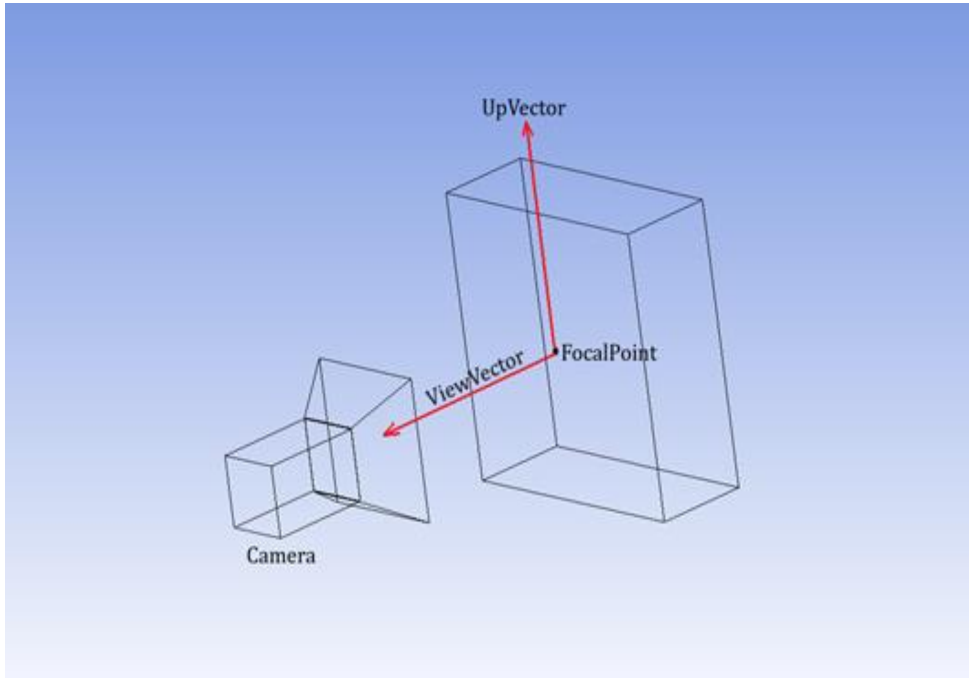


Mechanical Scripting API improvements - Camera

The WB Camera is analogous to a digital camera.
The visualization of the model can be manipulated by changing the properties and methods of the camera ACT.

The following properties together represent the state of the camera.

- FocalPoint: specifies the location on the object where the camera is looking at.
- ViewVector: direction the camera is looking at.
- UpVector: specifies the orientation (tilting) of the camera.
- ZoomFactor: a value of 1 will roughly fit the window. Larger values will make the model appear smaller while smaller values will make the model appear larger.



Scripting API - Camera (continued)

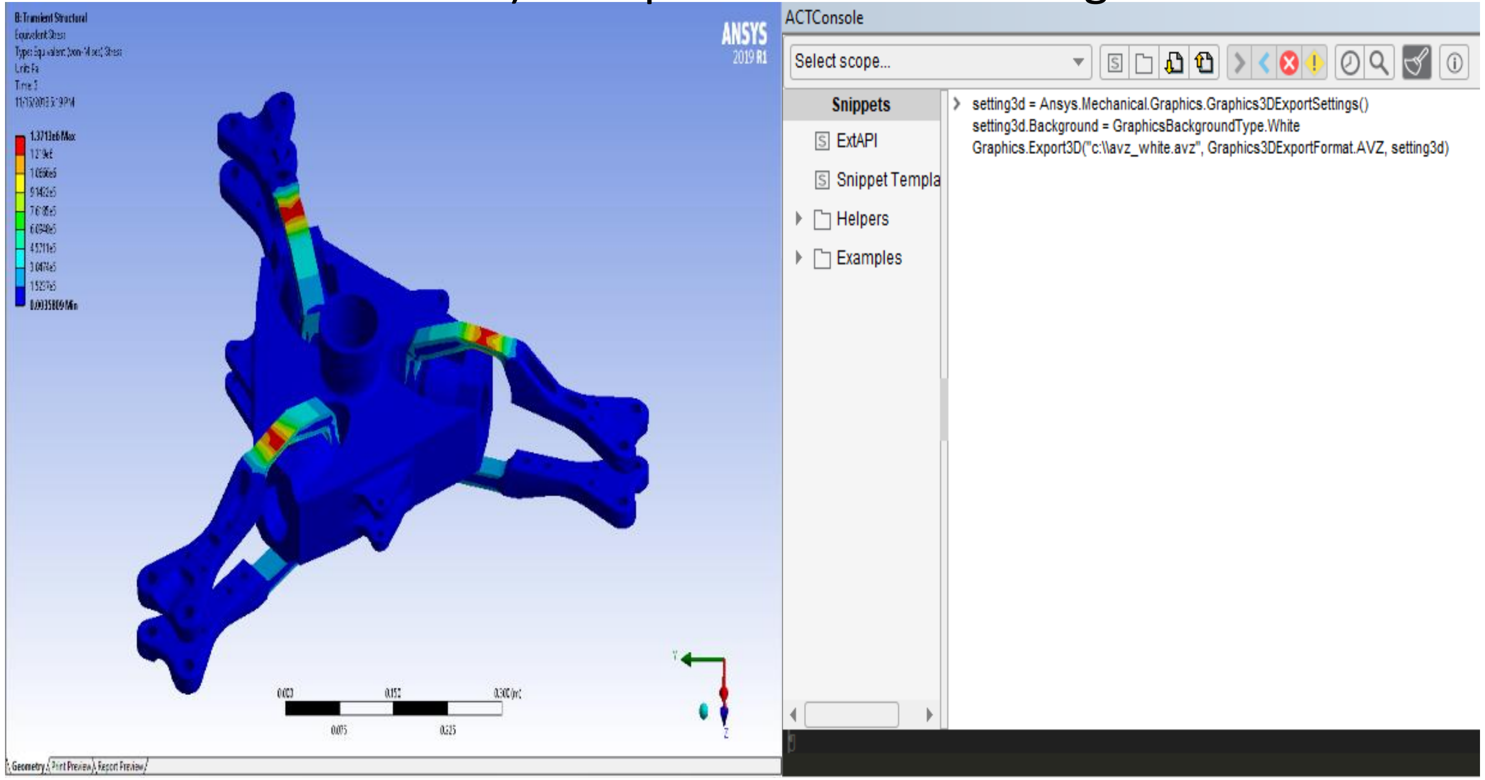
These methods are available to manipulate the camera in a more user-friendly manner:

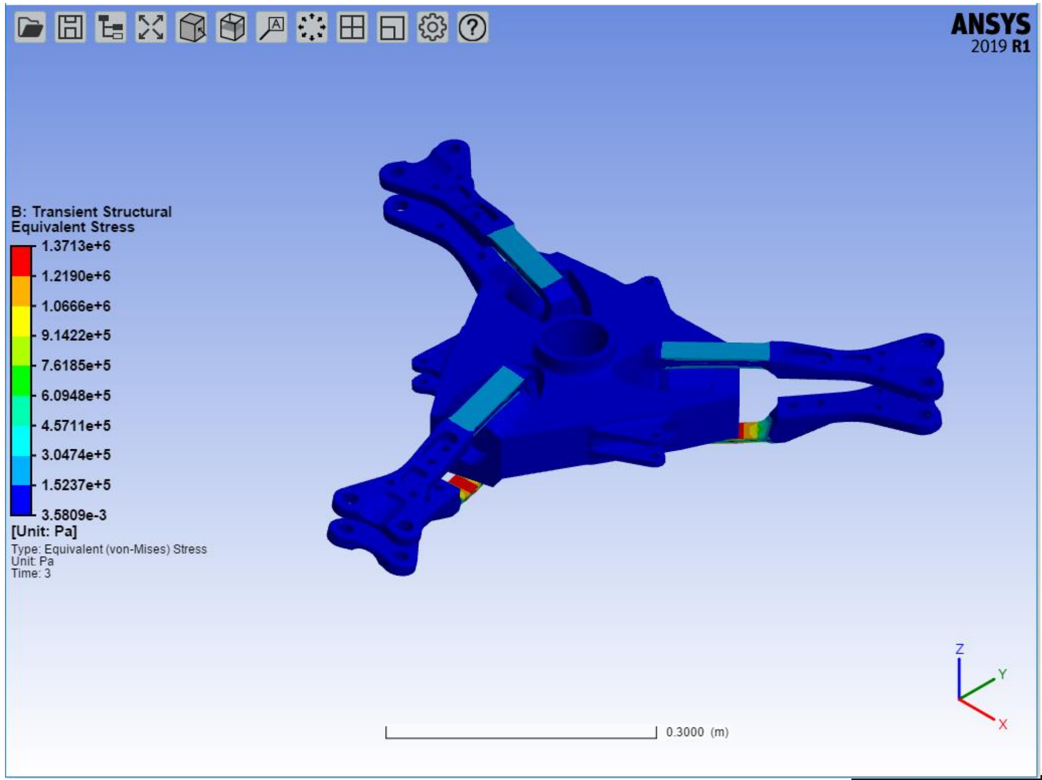
- SetFit: Fit the view to the whole model
- Rotate: Rotate along a particular axis
- SetSpecificViewOrientation: Set the orientation to one of the following predefined viewing orientations (with respect to the global Cartesian coordinate system):
    - **Front**: Front direction (i.e. 0, 0, 1)
    - **Back**: Back direction (i.e. 0, 0,-1)
    - **Top**: Top direction (i.e. 0, 1, 0)
    - **Bottom**: Bottom direction (i.e. 0,-1, 0)
    - **Left**: Left direction (i.e. -1, 0, 0)
    - **Right**: Right direction (i.e. 1, 0, 0)
    - **Iso**: Iso direction (i.e. 1, 1, 1)

Scripting API improvements - Export Graphics Display

The graphics display can be exported using the following properties

- Export3D method can be used to export the 3D model in STL and AVZ format.
- ExportImage exports the image to a PNG, JPG, TIF, BMP, or EPS file.
- Graphics3DExportSettings and GraphicsImageExportSettings lets modify settings (background, resolution etc.) to export model and image.





Scripting API improvements - Section Plane

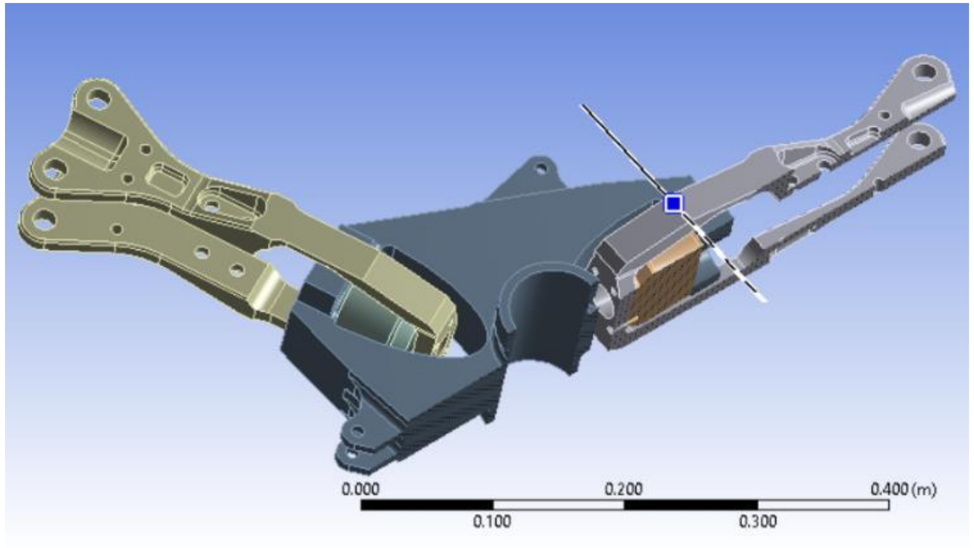
This allows the user to perform functionalities related to creating a slice or section plane on your model so that you can view internal geometry, mesh, and/or result displays.

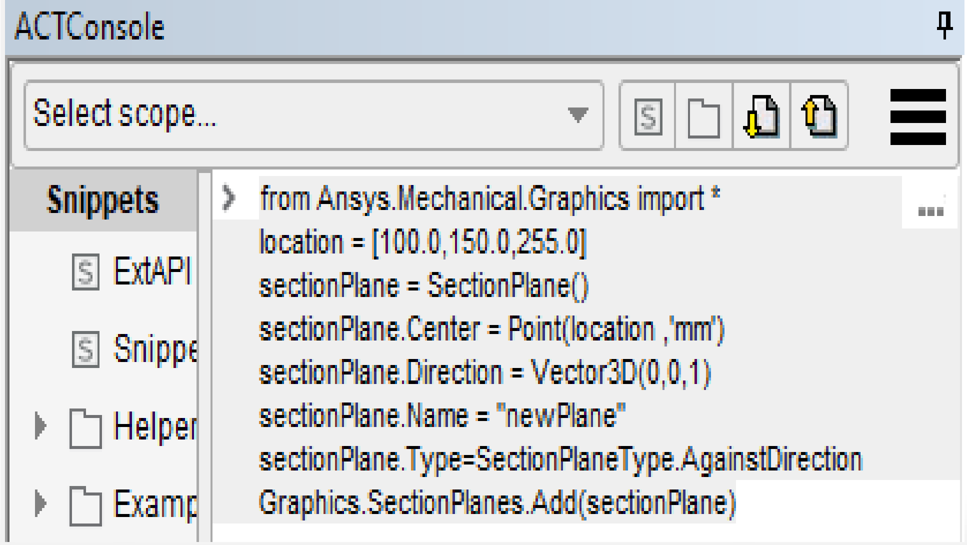
The following methods can be used to manipulate section planes:

- Add: adds a new section plane to the collection.
- Remove: remove a section plane from the collection.
- RemoveAt: remove a section plane at a given index.
- Clear: clears all section planes from the collection.

The following properties can be used to get/modify state of the section plane:

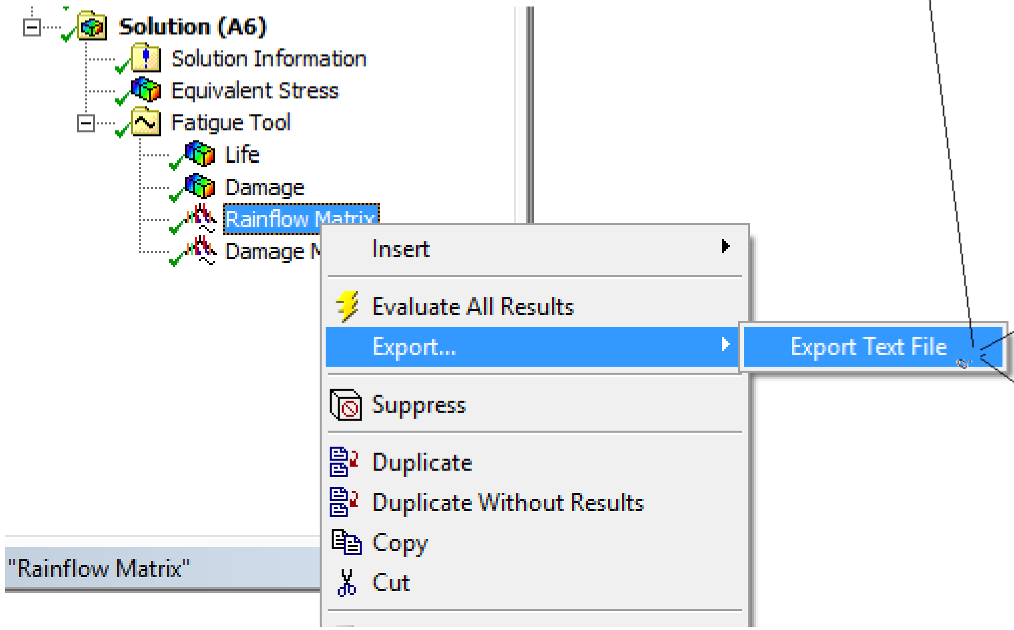
- **Capping**: capping style
- **ShowWholeElement**: element visibility of section plane
- **Center**: center point of a section plane
- **Type**: type of section plane
- **Name**: name of section plane
- **Active**: active state of section plane
- **Direction**: normal direction of section plane

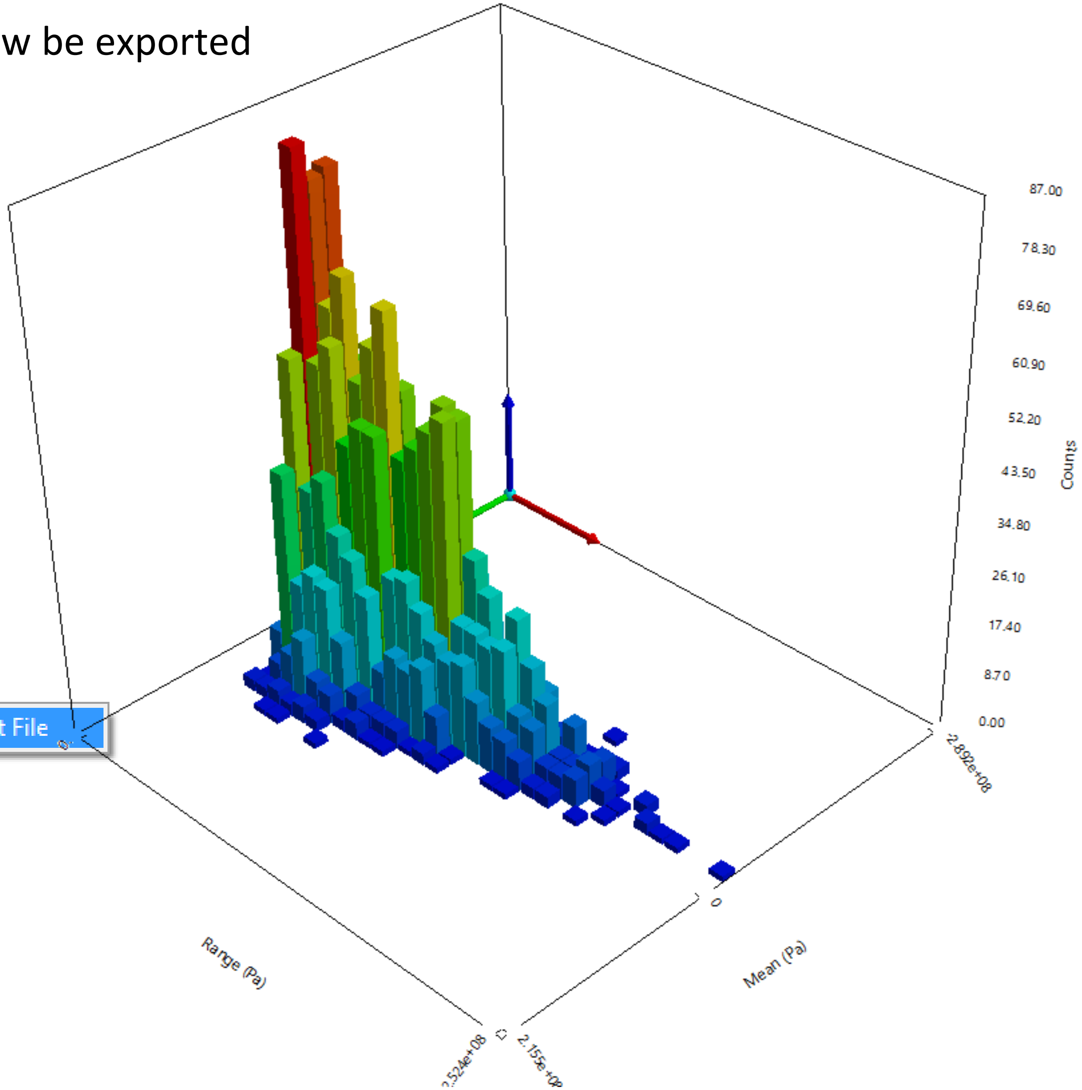




Fatigue results export

The Fatigue results Rainflow Matrix and Damage Matrix can now be exported as a text file





CMS in WB-Mechanical

ANSYS 2019R1 update

Outline

CMS exposure for MSUP harmonic response analysis

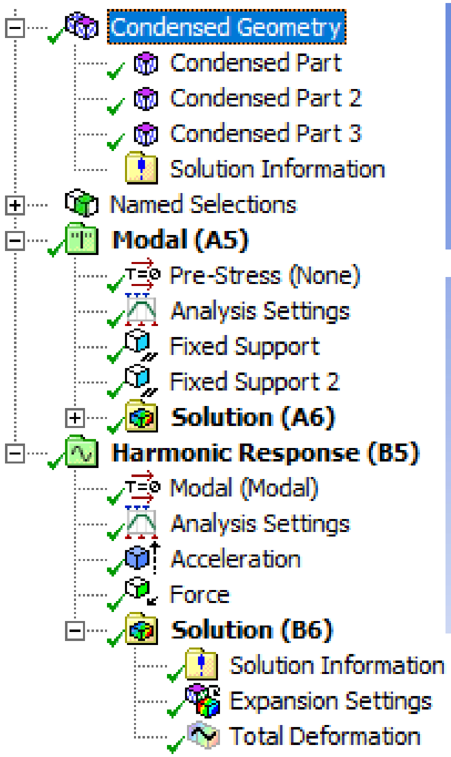
Bushing formulation exposure for Bushing joint which can be internal to Condensed Part

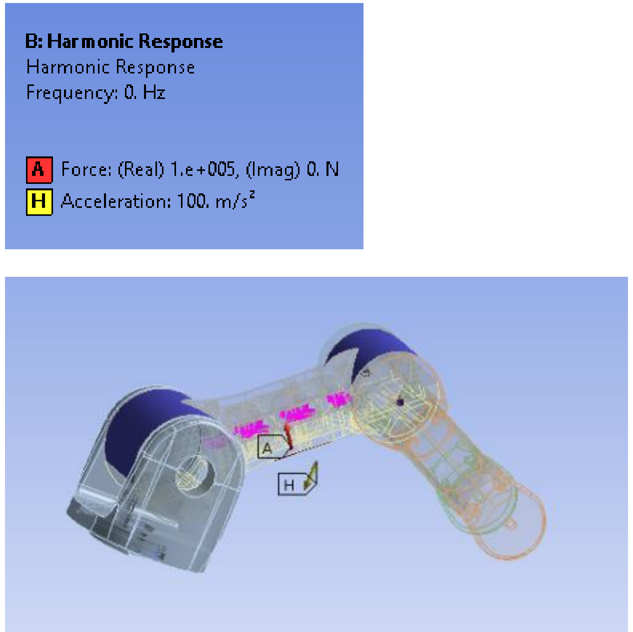
Support of Acceleration loads in MSUP harmonic using Substructure restart procedure

Improve disk space and performance by performing file reference for use pass and expansion pass instead of file copy

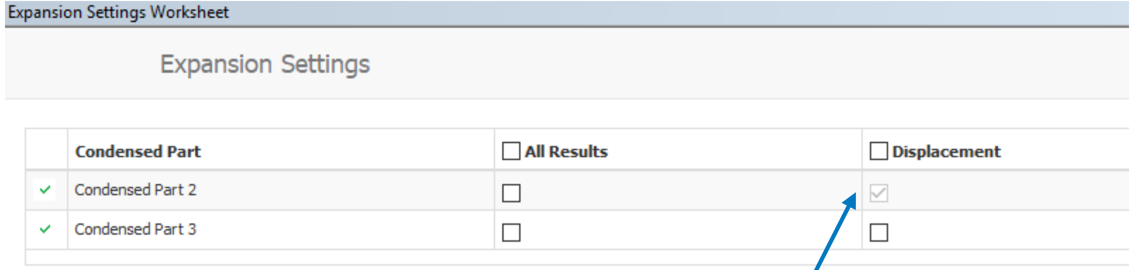
CMS exposure for MSUP harmonic analysis

Standalone and Linked MSUP Harmonic analysis can now use CMS based matrix reduction method to work with Substructures.





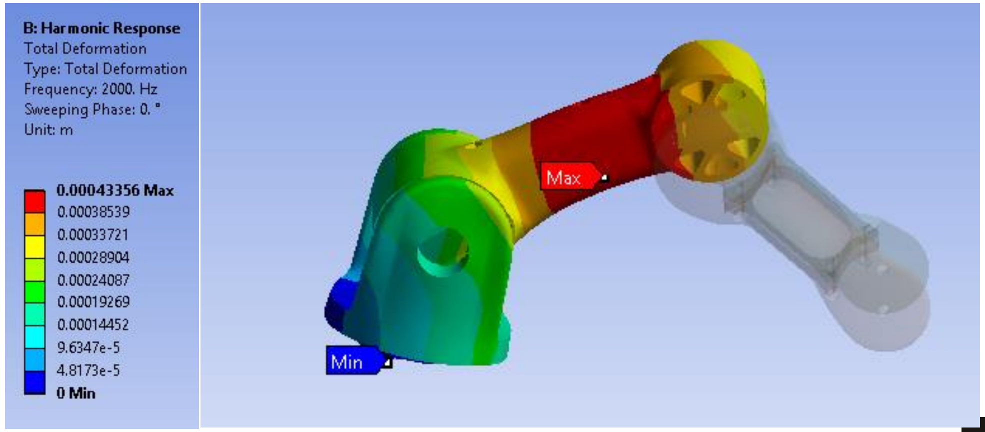
Graphics are made transparent for condensed part and MDOF is displayed on condensed parts


Expansion Settings Worksheet

Expansion Settings

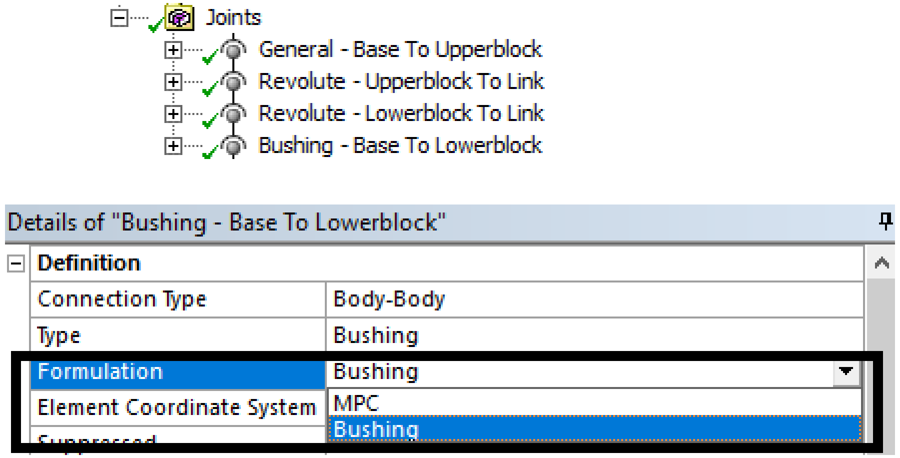
	Condensed Part	All Results	Displacement
✓	Condensed Part 2	☐	☑
✓	Condensed Part 3	☐	☐

Expansion enabled for Condensed Part 2 and seen in the deformation results

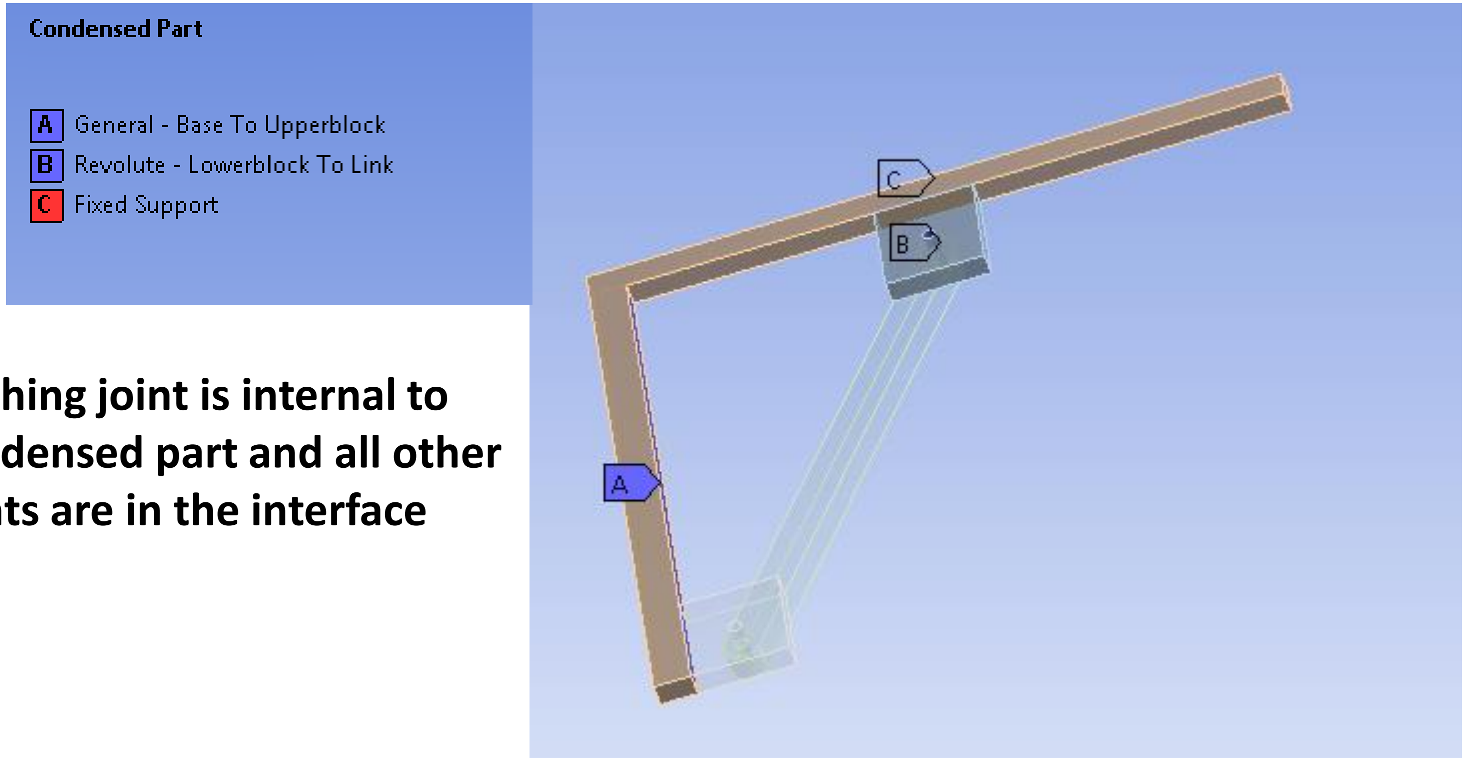


Bushing formulation for CMS based Modal + Harmonic

Bushing joint now supports Bushing formulation which can be internal to Condensed Part and can be included in Generation Pass for Modal and Harmonic analysis



Bushing joint is internal to condensed part and all other joints are in the interface



ANSYS

Acceleration load support for Harmonic analysis

CMS based MSUP Harmonic analysis supports Acceleration load. Since Acceleration is scoped to All Bodies in the analysis, it needs to be applied to both Condensed Part which are within substructures as well as Non-Condensed parts. To support the Acceleration load, substructure restart procedure is performed in Harmonic analysis during use pass.

```
/com,*********** Performing Substructuring Re-start ***********
/gopr
/clear
/FILNAME,cp133         ! Jobname = cp133
resume,cp133,db
/solu
antype,subs,restart    ! restarting the substructuring analysis
/com,*********** Create Acceleration ***********
acel,1,0,0                                ! generate reduced LV1
solve
acel,0,1,0                                ! generate reduced LV2
solve
fini
/clear
```


Improve disk space and performance

MODDIR is activated in the Super-element Generation pass of MAPDL solution. This enables the Use pass and Expansion pass to do file reference, when required. This avoids file copy of the LN22 files and other files shown below to improves disk space and performance due to file copy

```
cp128.rst
cp128.LN22/LN09/LN20
cp128.full
cp128.cms
cp128.emat
cp128.bcs
```

These files are read remotely through file reference

```
cp128.sub (c)
cp128.bclv (c)
cp128.seld/mlv (c)
cp128.db (c)
cp128.esav (c)
```

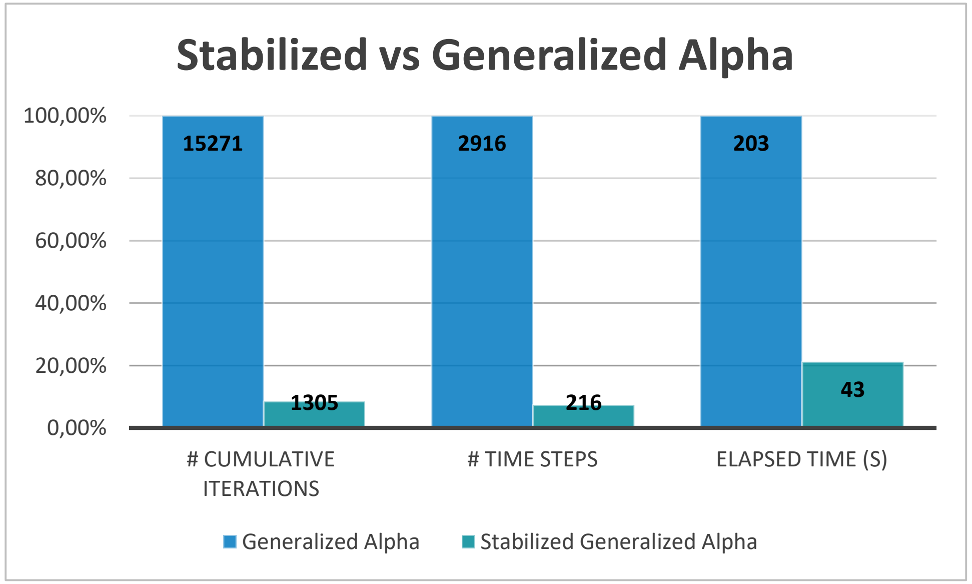
These files are copied from condensed part folder during use pass and expansion pass

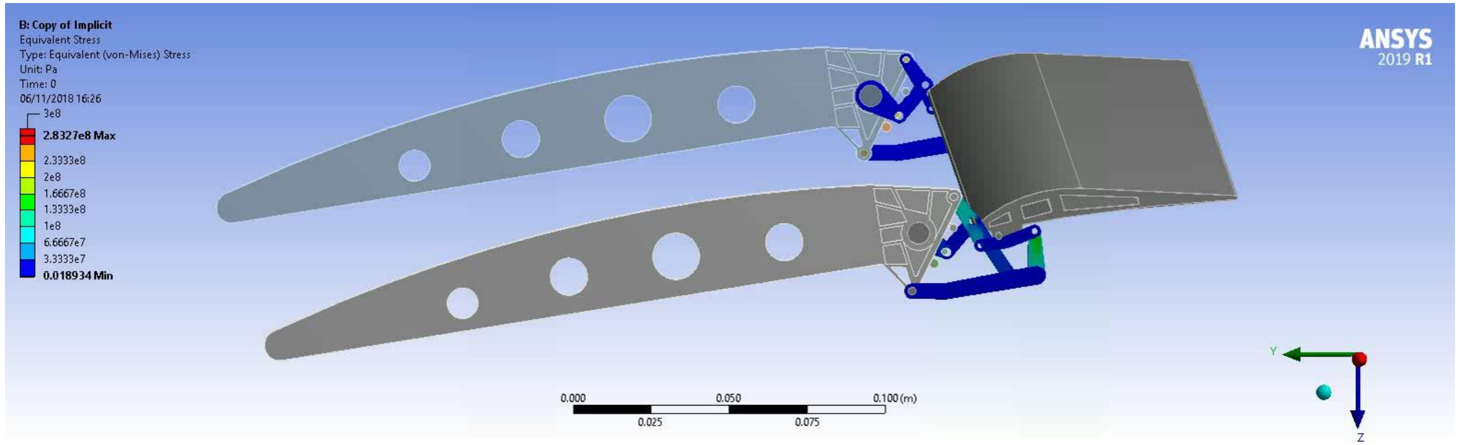
Rigid Body Dynamics

ANSYS 2019 R1 update

Stabilized Generalized Alpha time integration

- A posteriori correction of constraints leads to high frequencies oscillations that needs ➔ penalizes NR convergence and timesteps
- New Stabilized Generalized Alpha time integration enforces constraints while preserving dynamic equilibrium ➔ no spurious oscillation ➔ faster convergence and larger time steps



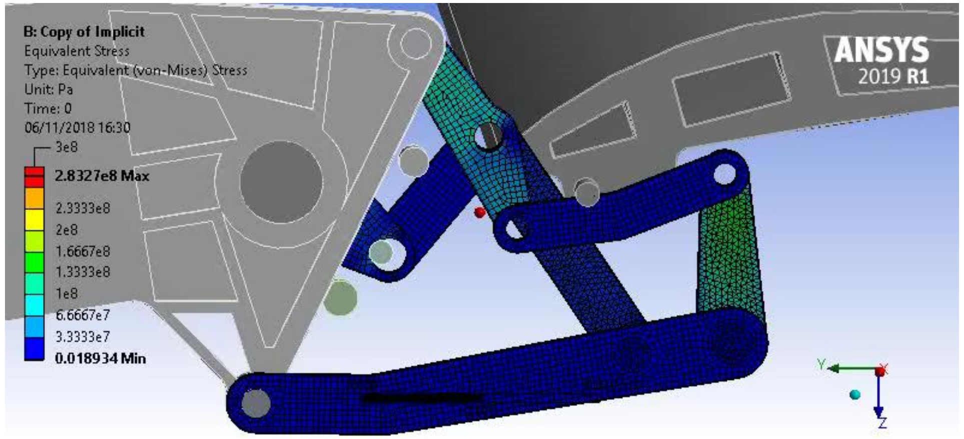


On-demand CMS expansion

- CMS expansion may lead to large RST files with large meshes and/or large number of time points (transient simulation) which are expensive to write/read

➔ Perform expansion on the fly during post to avoid any file

➔ Based on the Data Processing Framework



Medium size model: 14 Condensed Parts, 165867 nodes, 60859 elements, 102 time points

- **MAPDL expansion: 4ms16, 14 RST files, total: 3.5 Gb**
- **On-demand expansion: 22s (deformation+stress), no RST file**

Miscellaneous

- Worksheet to review step aware analysis settings
- Ability to define restitution factor as a parameter
- Command to output detailed contact forces
- Command to print body force balance
- 8 Customer Defects addressed.

Worksheet

Analysis Settings

Properties	Step 1	Step 2	Step 3	Step 4
Step Controls				
Step End Time	0.1	0.5	1.	5.
Auto Time Stepping	Off	On	On	On
Carry Over Time Step	N/A	Off	On	On
Time Step	1.e-002	N/A	N/A	N/A
Initial Time Step	N/A	1.e-003	N/A	N/A
Minimum Time Step	N/A	1.e-007	1.e-007	1.e-007
Maximum Time Step	N/A	5.e-003	5.e-002	5.e-002
Output Controls				
Store Results At	All Time Points	All Time Points	All Time Points	Equally Spaced Points
--- Value	N/A	N/A	N/A	100

Details of "Frictionless - A0 To B0"

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	A0
Target Bodies	B0
Protected	No
Definition	
Type	Frictionless
Advanced	
Pinball Region	Program Controlled
P Restitution Factor	1
RBD Contact Detection	Program Controlled

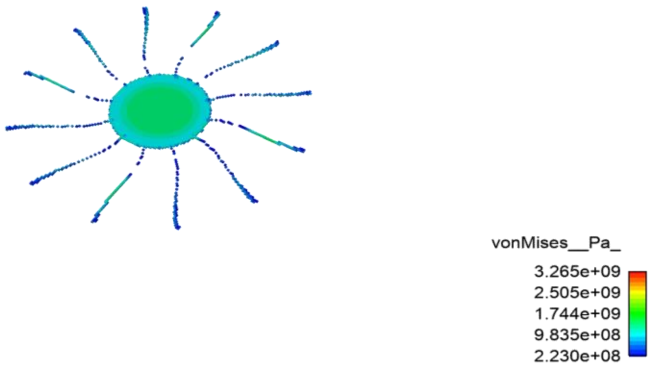
Details of "Revolute - B01 To B1"

Reference	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	2 Faces
Body	B01
Coordinate System	Reference Coordinate System
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	2 Faces
Body	B1
Initial Position	Unchanged
Behavior	Rigid
Pinball Region	All
Stops	
RZ Min Type	Stop
RZ Min	0. °
RZ Max Type	None
P Restitution	1.

Linear Dynamics

Cyclic Symmetry Analysis – Initial State

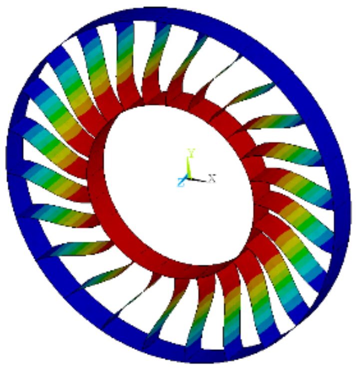
Objective
Include non-zero initial stress or strain in a cyclic symmetry analysis
Feature
Support INISTATE command (DTYP = STRE or EPEL)
Example Scenario: Jet Engine Simulation
• Read in the initial state for room temperature and impose it on a cyclic symmetry sector model. The initial stress/strain is, for example, caused by machining process. • Specify the temperature profile and rotational speed at the engine condition. Run a prestress static analysis. • Run a linear perturbation cyclic symmetry modal analysis.



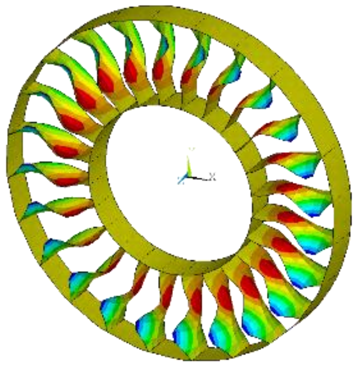
Residual stresses from additive manufacturing is an example of an initial stress state

Cyclic Symmetry Analysis – Damped Modal Analysis

Objective
Include damping and/or Coriolis effect in a cyclic symmetry modal analysis
Feature
Support damped eigensolvers (MODOPT,DAMP and QRDAMP)



Real Mode



Imaginary Mode

Complex Mode

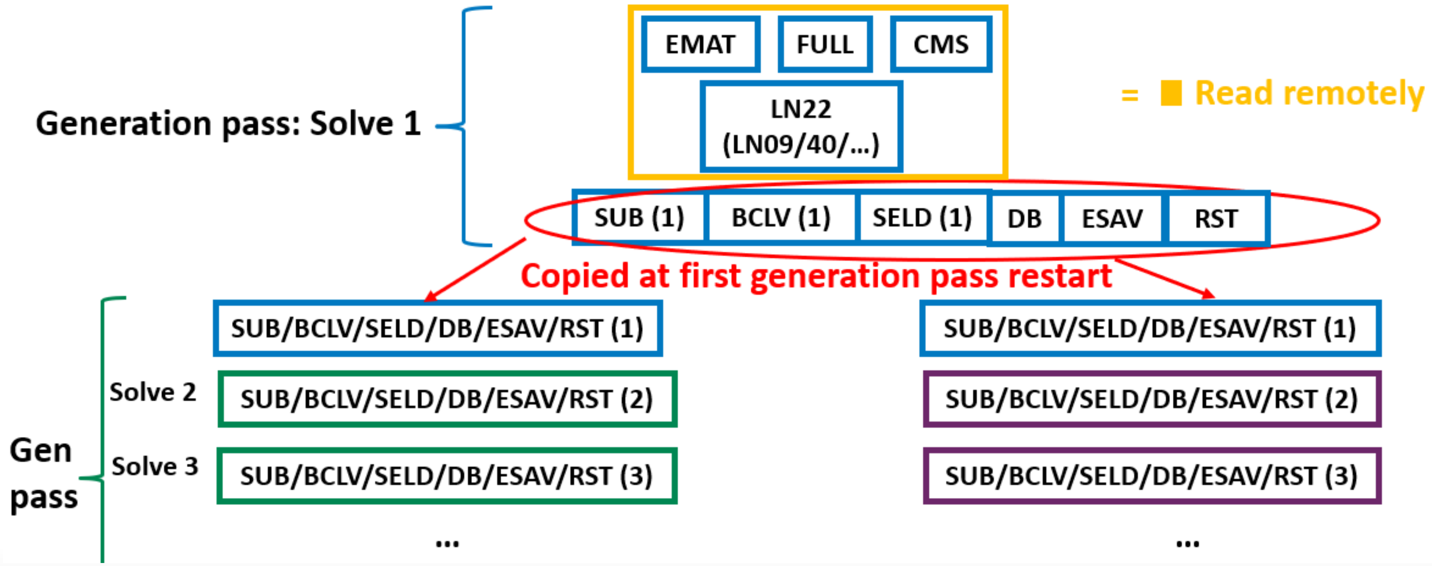
***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ(Damped)			TIME/FREQ(Undamped)	LOAD STEP	SUBSTEP	CUMULATIVE	HRM-INDEX
(49,50)	-0.74981	103.28	j	103.28	3	1	25	2
(25,26)	-0.75499	103.68	j	103.68	2	1	13	1
(27,28)	-0.75499	103.68	j	103.68	2	2	14	1
(1, 2)	-0.83780	109.85	j	109.85	1	1	1	0
(29,30)	-1.0441	123.89	j	123.90	2	3	15	1
(31,32)	-1.0441	123.89	j	123.90	2	4	16	1
(51,52)	-1.0583	124.80	j	124.81	3	2	26	2
(3, 4)	-1.2854	138.53	j	138.54	1	2	2	0
(33,34)	-4.8091	274.32	j	274.36	2	5	17	1
(35,36)	-4.8091	274.32	j	274.36	2	6	18	1
(5, 6)	-6.3740	316.45	j	316.51	1	3	3	0
	damping	oscillation						

Complex frequencies

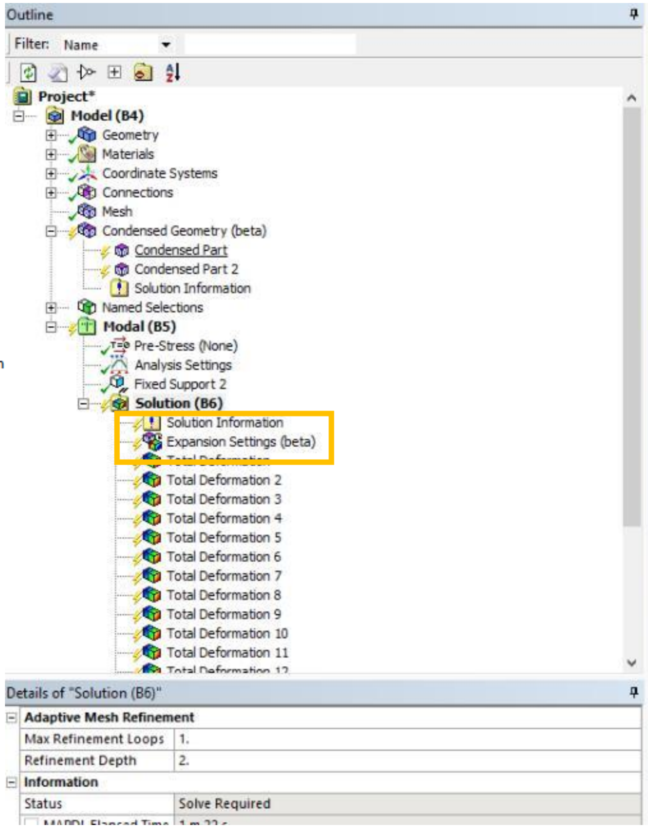
Substructuring / CMS – Remote Files Usage

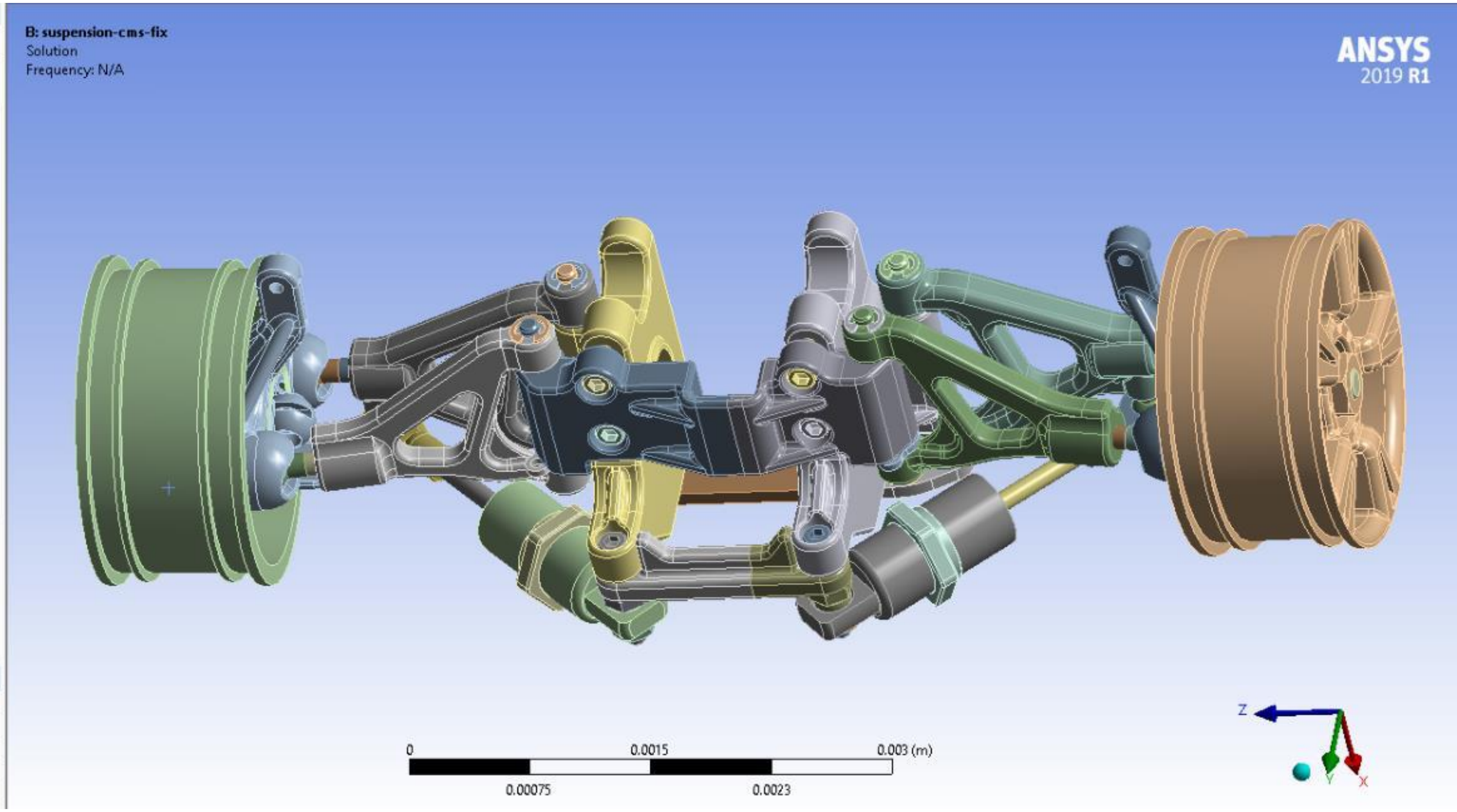
Objective
Optimize substructuring/CMS file handling for Mechanical scenarios. Avoid copies. Access files remotely.
Feature
Support MODDIR command during the first solution of the first restart of a generation pass, or during the first solution of the expansion pass.



Substructuring / CMS – Remote Files Usage

Application: Mechanical Scenario

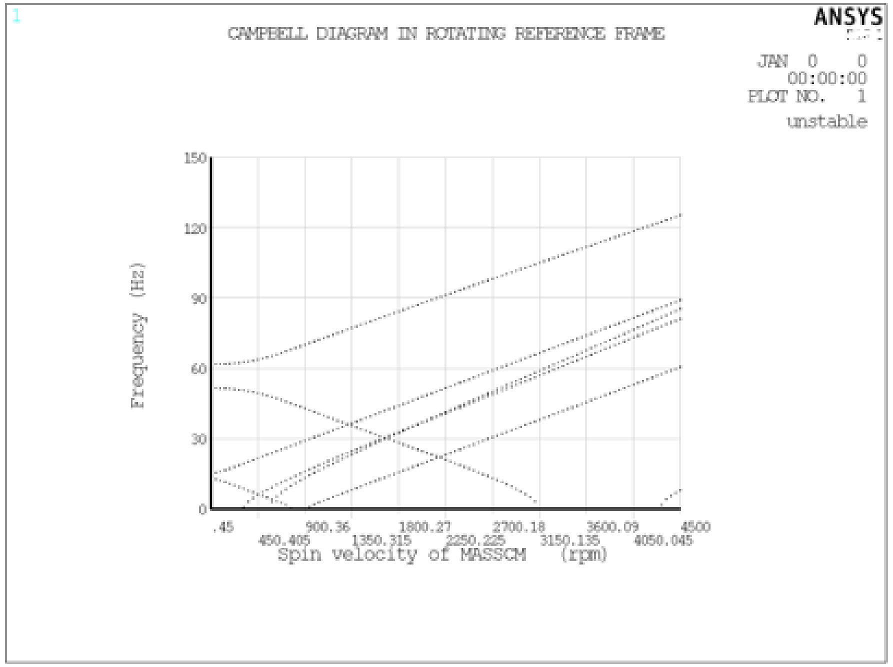





```
! ANSYS input file written by Workbench
/filn,cp914
resume
/solu
expass,on
outres,erase
outres,all,none
outres,nsol,all
outres,rsol,all
outres,strs,all
outres,eangl,all
outres,fmag,all
outres,curd,all
outres,nldat,all
outres,eheat,all
outres,etmp,all
outres,srfs,all
outres,cont,all
outres,bkstr,all
outres,veng,all
outres,epel,all
seexp,cp914,use
numexp,all,,,No
modd,on,'C:\Users        \AppData\Local\Temp\AFT_R191_suspension.tmp\AFT_R191_suspension_files\dp0\global\MECH\SYS-1\Condensed Geometry\cp914\',cp914
solve
fini
/gopr
*get,_walldone,active,,time,wall
_preptime=(_wallbsol-_wallstrt)*3600
_solvtime=(_wallasol-_wallbsol)*3600
_posttime=(_walldone-_wallasol)*3600
_totaltim=(_walldone-_wallstrt)*3600
/wb,file,end               ! done with WB generated input
```

Rotating Structure Analysis - Rotating Reference Frame Analysis

Objective
Extend the applicability and improve the accuracy of the rotating reference frame analyses
Features
When activating CORIOLIS,ON: • Coriolis and spin-softening on rotational degrees of freedom • SYNCHRO procedure • Campbell procedure • Rotating damping effect • Periodic transient forces (COMBI214)
Documentation
Enhanced and cleaned-up dedicated part in Advanced Analysis Guide. 3 new Verification Manual examples.



Rotating Structure Analysis - Rotating Reference Frame Analysis

Rotating Machine Dynamics

Linearization to support linear analyses: modal, harmonic, and linear transient

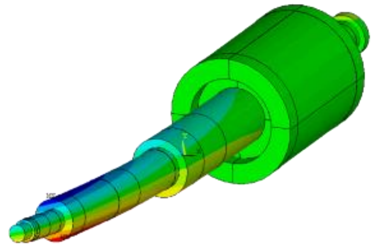
Axisymmetric possibly multi-body

One body on orthotropic supports

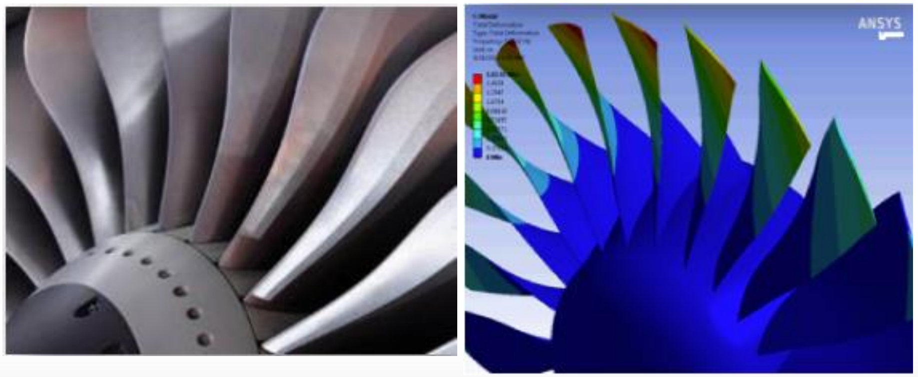
Full non-linear transient analysis

No limitation on the model

Stationary Reference Frame Analysis = Rotordynamics analysis (MAPDL Rotordynamics Guide, some Mechanical exposure)

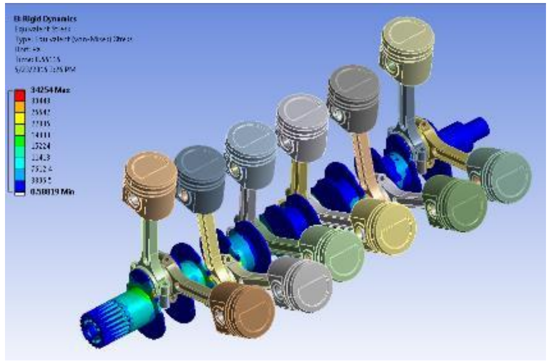


Rotating Reference Frame Analysis in 2019R1 (MAPDL Advanced Analysis Guide)



Co-rotational Reference Frame - Flexible multi-body analysis:

- RBD
- MAPDL



Rotating Structure Analysis - Rotating Reference Frame Analysis – Verification Manual Examples

VM197

Rotating Elastic System

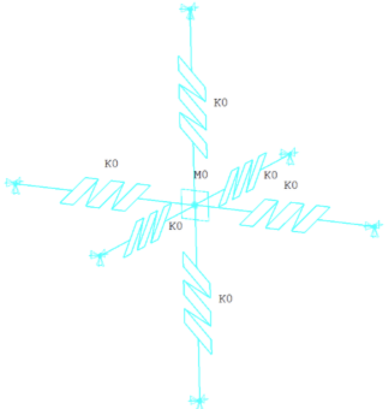
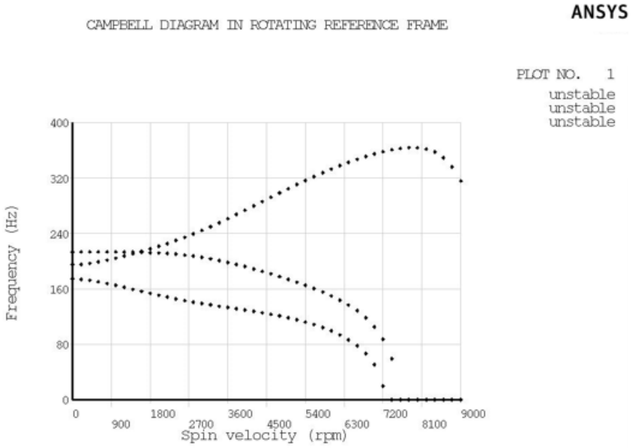
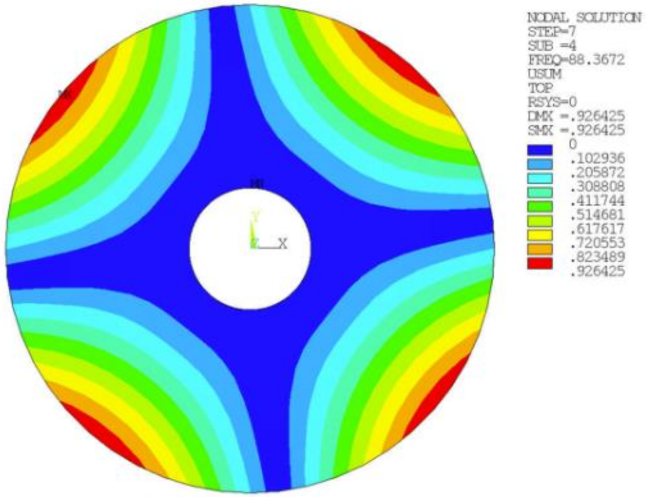


Figure 303: Campbell Diagram – Case 2



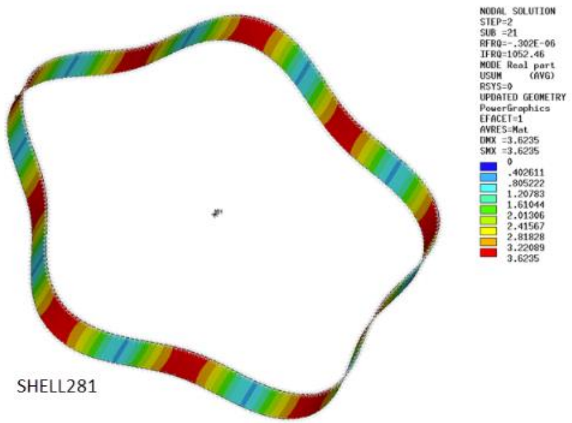
VM301

Critical Speed of a Rotating Disk

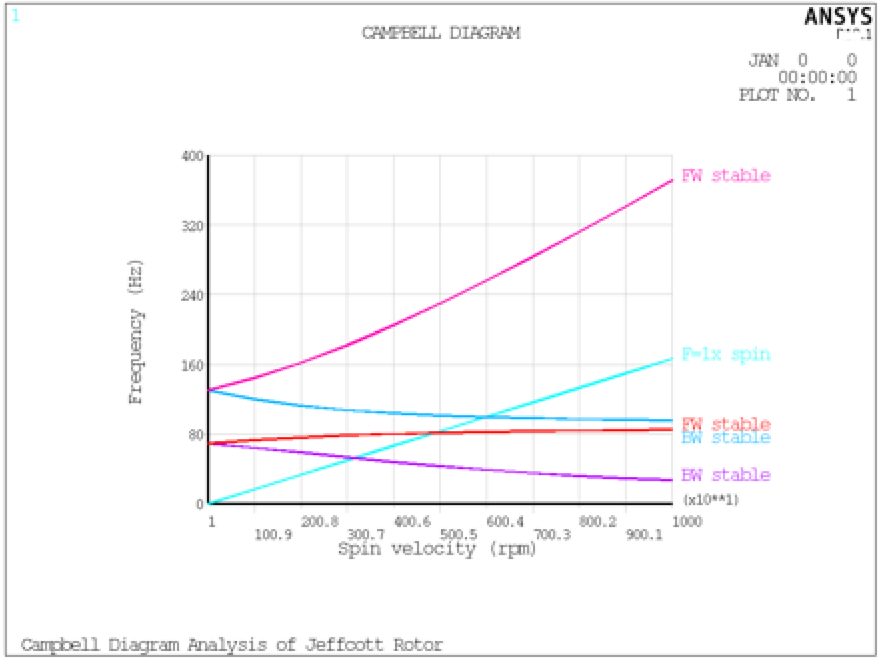


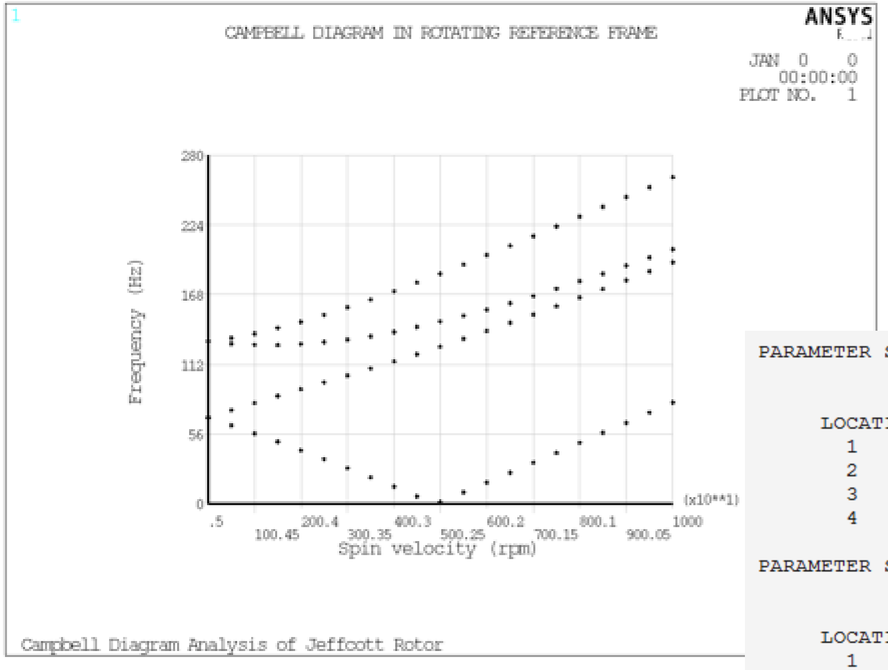
VM302

Rotating Circular Ring



Rotating Structure Analysis - Rotating Reference Frame Analysis – Jeffcott Rotor

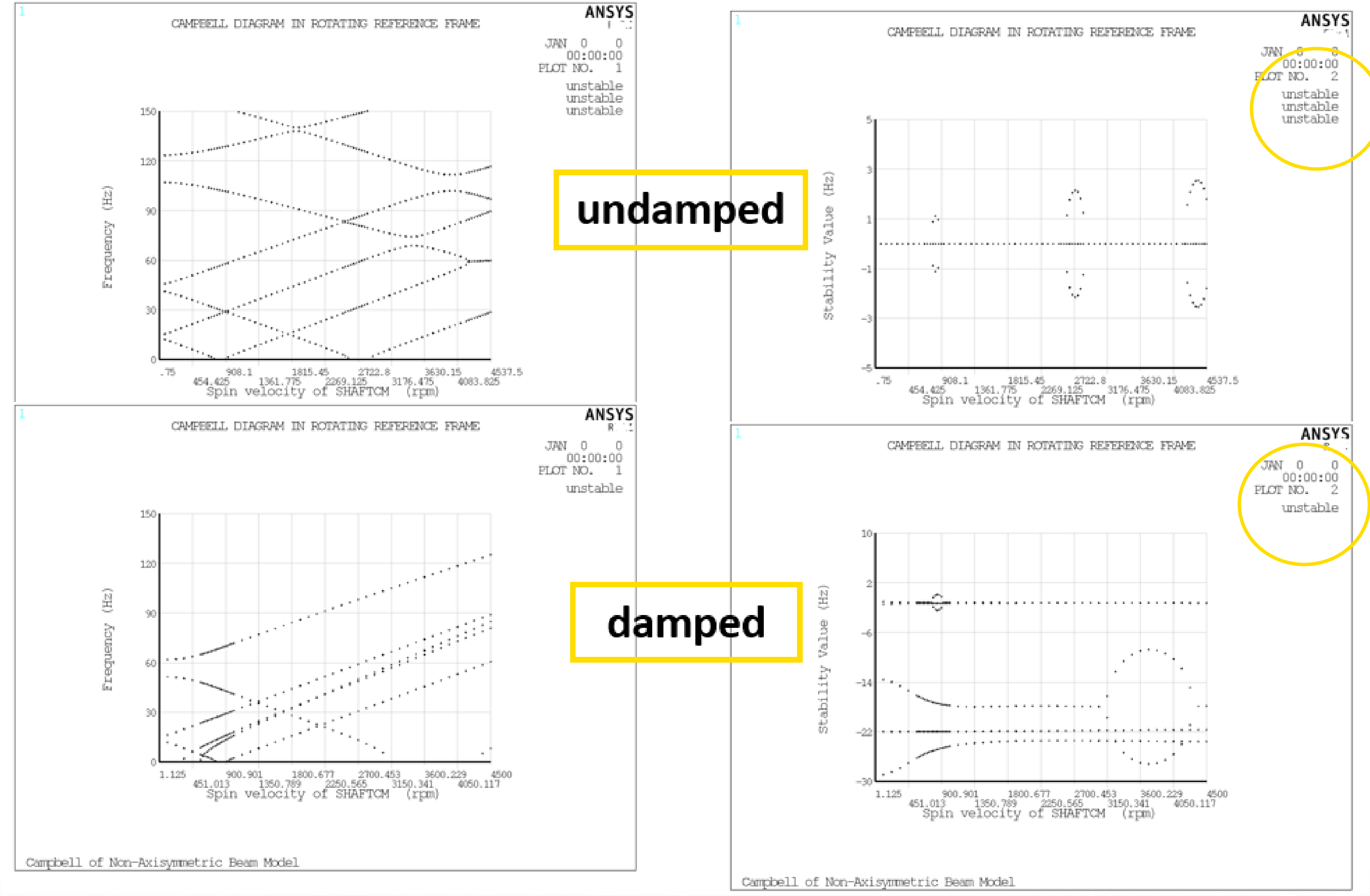





```
PARAMETER STATUS- FRQ_SRF  (     21 PARAMETERS DEFINED)
                  (INCLUDING        2 INTERNAL PARAMETERS)

     LOCATION                VALUE
        1         1        1     27.1023115
        2         1        1     85.1999476
        3         1        1     95.8462296
        4         1        1     371.081927

PARAMETER STATUS- FRQ_RRF  (     21 PARAMETERS DEFINED)
                  (INCLUDING        2 INTERNAL PARAMETERS)

     LOCATION                VALUE
        1         1        1     81.4667190
        2         1        1     193.768978
        3         1        1     204.415260
        4         1        1     262.512896

PARAMETER STATUS- FRQ_SRF_INRRF  (     21 PARAMETERS DEFINED)
                  (INCLUDING        2 INTERNAL PARAMETERS)

     LOCATION                VALUE
        1         1        1     81.4667190
        2         1        1     193.768978
        3         1        1     204.415260
        4         1        1     262.512896
```

±Ω relationship between stationary reference frame and rotating reference frame frequencies is verified

Rotating Structure Analysis - Rotating Reference Frame Analysis – Stability and Damping Effect



Asymmetric shaft (BEAM188) with disks (MASS21) on bearings (COMBI214)

2019 R1 Linear Dynamics, CMS, Acoustics & NVH

ANSYS 2019R1 update

Outline

1. Material dependent damping based on Constant Structural Damping Coefficient
2. Bushing formulation (COMBI250) for Modal and Harmonic analysis
3. Participation factor output for Modal analysis
4. Command snippet enhancement to exclude solve command for a particular step or number of steps
5. CMS
6. Acoustics & NVH

Material based Constant Structural Damping Coefficient

1. Engineering data supports Material Dependent Damping group, which consists of two properties Damping Ratio and Constant Structural damping coefficient.
2. Constant structural damping coefficient is supported in 2019 R1 for Full Harmonic, Fully damped Modal, Reduced Damped Modal (When complex solution is set to Yes) and Full transient analysis.
3. The default of Constant Structural Damping Coefficient is set to twice of that of Damping Ratio specified by the user

Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m^-3		
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Material Dependent Damping				
7	Damping Ratio	0.1			
8	Constant Structural Damping Coefficient	= 0.2			
9	Isotropic Elasticity				

ANSYS

Material based Constant Structural Damping Coefficient

1. In current release, Constant Damping Coefficient in Engineering data has been renamed to Damping Ratio
2. The naming convention for these properties are made consistent across MAPDL and Mechanical and also within Analysis Settings and Engineering data in Mechanical
3. If Damping ratio is specified for the material on database resumed prior to 2019 R1 release, then it will add Material Dependent Constant Structural Damping Coefficient and will make its value twice of Damping ratio
4. Material Dependent Damping Ratio is no longer applicable for Full Harmonic analysis
5. Material Dependent Damping based on Damping ratio is sent as MP,DMPR command and Material Dependent Damping based on Constant Structural Damping Coefficient is sent to the solver as MP,DMPS command. Please see this link for supported analysis

Bushing formulation (COMBI250) for Modal + Harmonic

Bushing formulation is now supported for Bushing joints in Modal and Harmonic analysis (COMBI250 element). This formulation supports element coordinate system and only diagonal terms can be specified for Stiffness and Damping coefficients. It can be used as internal to Condensed Part for CMS analysis. In prior releases, only MPC formulation was used behind the scene (formulation property is new)

Details of "Bushing - Base To Lowerblock"

Definition	
Connection Type	Body-Body
Type	Bushing
Formulation	Bushing
Element Coordinate System	MPC Bushing

Details of "Bushing - Base To Lowerblock"

Definition	
Connection Type	Body-Body
Type	Bushing
Formulation	Bushing
Element Coordinate System	Coordinate System
Suppressed	No
Element APDL Name	
Reference	
Mobile	

Worksheet

Bushing - Base To Lowerblock

Stiffness Coefficients

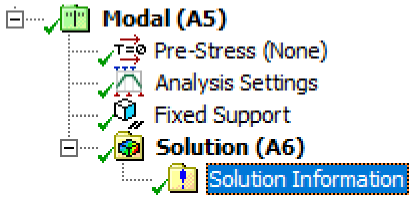
Stiffness	Per Unit X (m)	Per Unit Y (m)	Per Unit Z (m)	Per Unit θx (°)	Per Unit θy (°)	Per Unit θz (°)
Δ Force X (N)	10000					
Δ Force Y (N)		10000				
Δ Force Z (N)			10000			
Δ Moment X (N·m)				174.53		
Δ Moment Y (N·m)					174.53	
Δ Moment Z (N·m)						174.53

Damping Coefficients

Viscous Damping	Per Unit X (m)	Per Unit Y (m)	Per Unit Z (m)	Per Unit θx (°)	Per Unit θy (°)	Per Unit θz (°)
Δ Force * Time X (N·s	10.					
Δ Force * Time Y (N·s		10.				
Δ Force * Time Z (N·s			10.			
Δ Moment * Time X (				0.17453		
Δ Moment * Time Y (					0.17453	
Δ Moment * Time Z (						0.17453

Participation factor for Modal analysis

Participation factor summary is now supported for 2D Modal analysis



Details of "Solution Information"

Solution Information	
Solution Output	Participation Factor Summary
Summary Type	Ratio of Effective Mass to Total Mass
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	

Participation Factor Summary

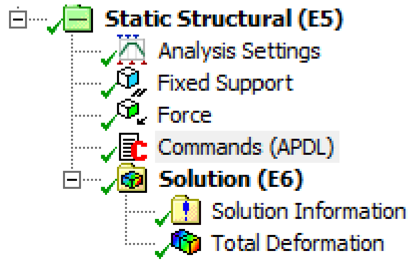
Ratio of Effective Mass to Total Mass

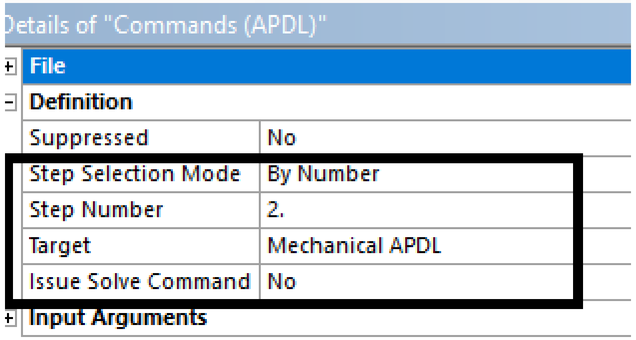
Mode	Frequency [Hz]	X Direction	Y Direction	Z Direction	Rotation X	Rotation Y	Rotation Z
1	250.66	1.4878e-032	1.4536e-033	---	---	---	6.8699e-032
2	284.11	1.7018e-032	1.5673e-032	---	---	---	4.1905e-033
3	433.9	4.8859e-032	1.6899e-032	---	---	---	2.003e-032
4	479.54	4.7643e-030	3.3906e-030	---	---	---	9.7104e-031
5	740.28	2.008e-023	1.0429e-024	---	---	---	0.8202
6	821.87	0.18719	0.12571	---	---	---	9.0954e-024
Sum		0.18719	0.12571	0.	0.	0.	0.8202

NOTE: The data displayed in the current worksheet is with respect to the solver unit system.

Commands enhancement for Issuing solve command

Commands object now supports new property Issue Solver Command. This property when set to No for applicable steps using Step Selection Mode, then the solve command is not issued for those load steps. In the example below the solve is skipped for second load step and can be seen from the results

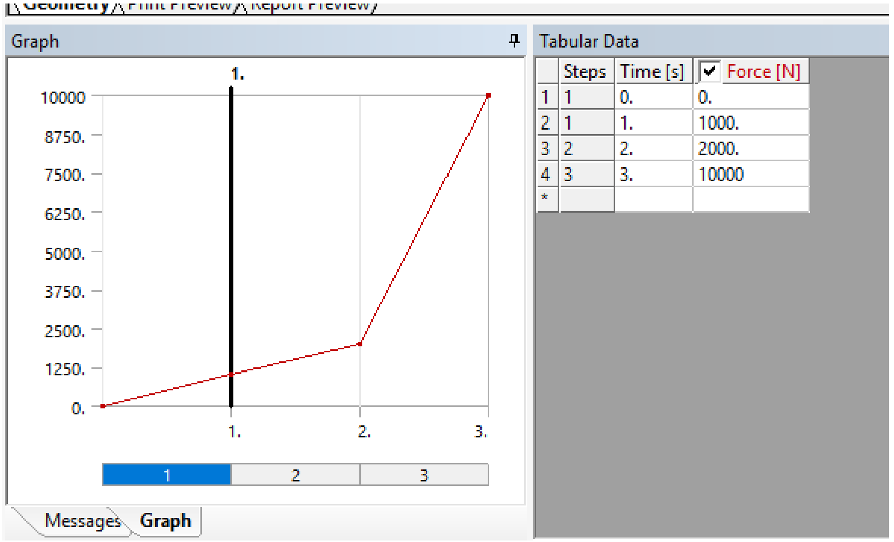




Details of "Commands (APDL)"

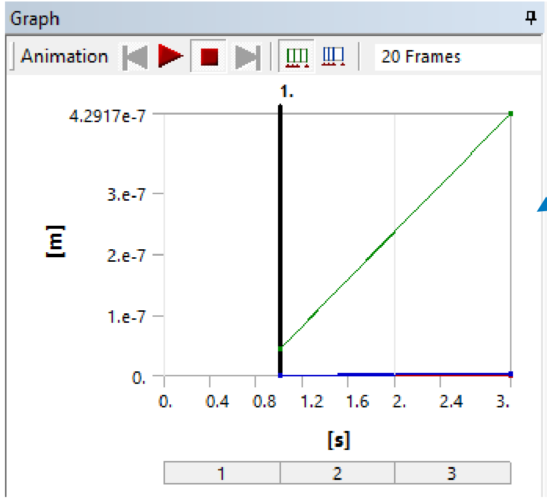
File	
Definition	
Suppressed	No
Step Selection Mode	By Number
Step Number	2.
Target	Mechanical APDL
Issue Solve Command	No
Input Arguments	

Loading

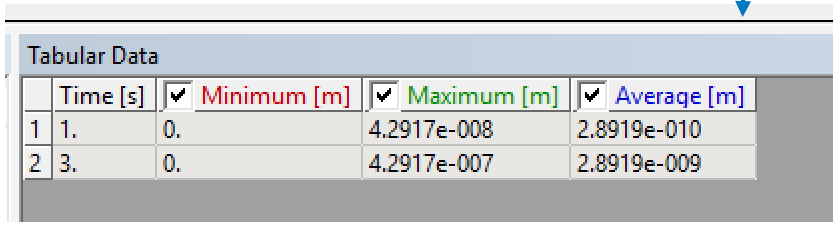


Tabular Data

	Steps	Time [s]	Force [N]
1	1	0.	0.
2	1	1.	1000.
3	2	2.	2000.
4	3	3.	10000
*			



Total deformation Graph and Tabular data



Tabular Data

	Time [s]	Minimum [m]	Maximum [m]	Average [m]
1	1.	0.	4.2917e-008	2.8919e-010
2	3.	0.	4.2917e-007	2.8919e-009

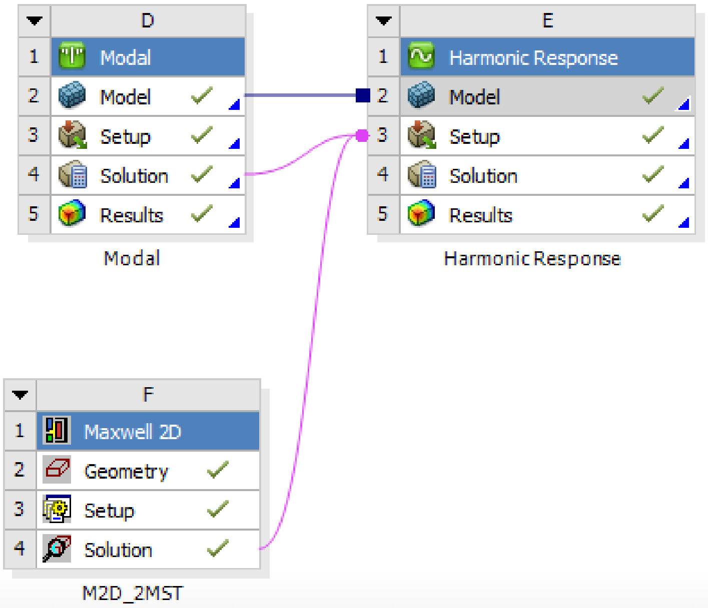
Acoustics & NVH

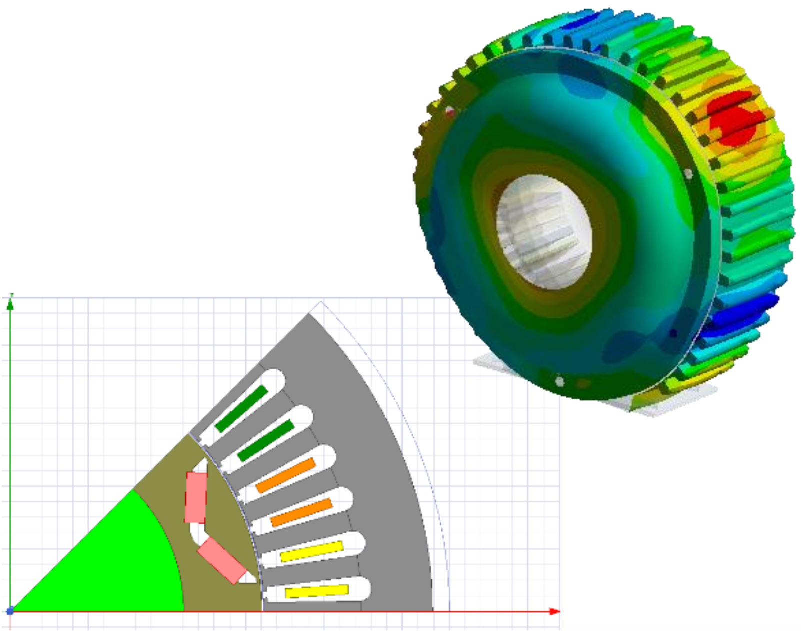
ANSYS 2019R1 update

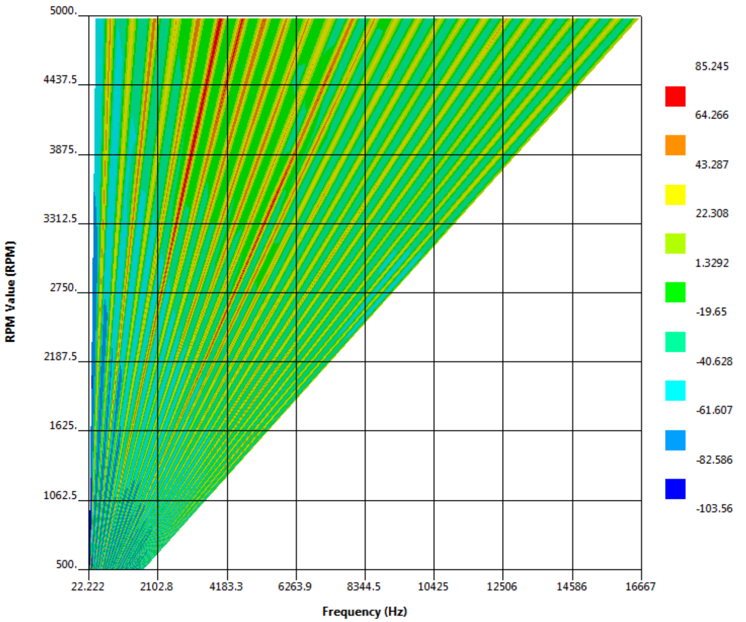
Maxwell-Mechanical: Multiple RPMs & ERP Waterfall Diagram

For Electric Machine design, it is important to analyse the acoustic signature of the system. In that goal, Equivalent Radiated Power can be calculated for a range of Rotational Velocities and Frequencies. We have develop a fully automated workflow to reach this goal.

- ✓ Perform DX Parametric study in Maxwell
- ✓ Transfer EMAG forces to Mechanical for all RPMs
- ✓ Plot ERP Waterfall Diagram



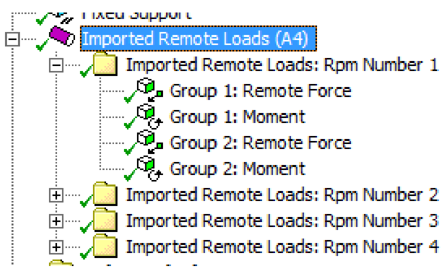


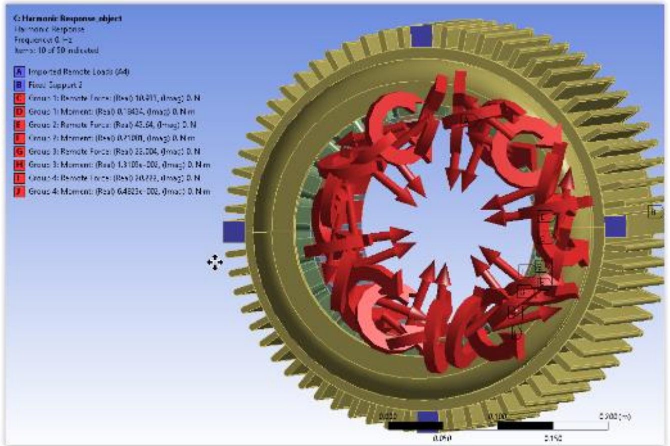


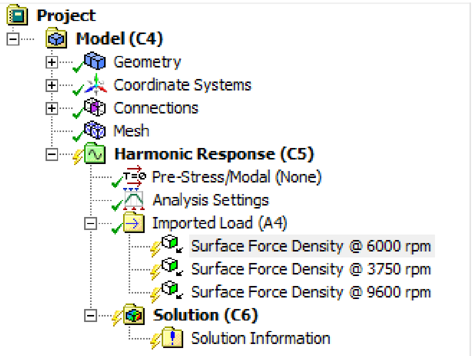
Maxwell-Mechanical: Multiple RPMs & ERP Waterfall Diagram

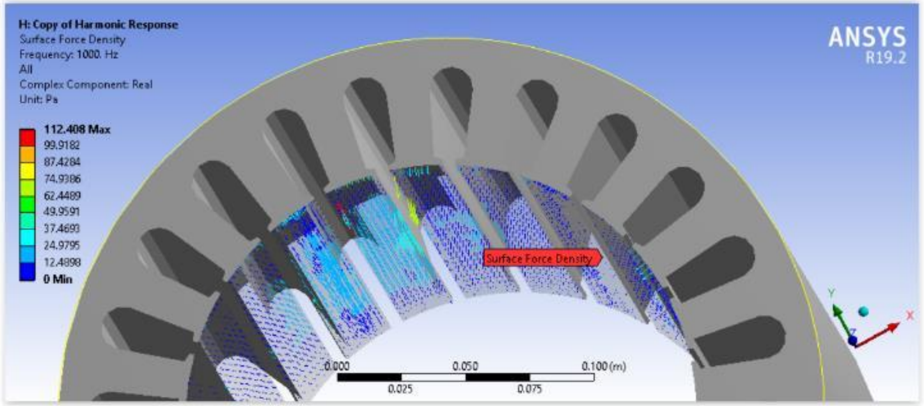
Two mapping strategies available depending on the geometry compliance:

- ✓ Object Based: Integrated Forces / Moments
- ✓ Mesh Based: Surface Force Densities



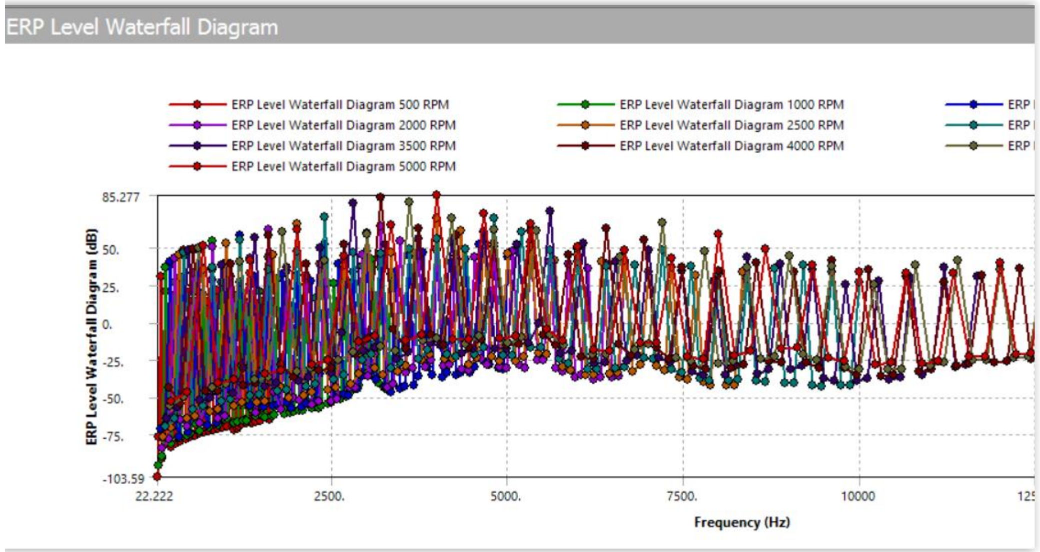


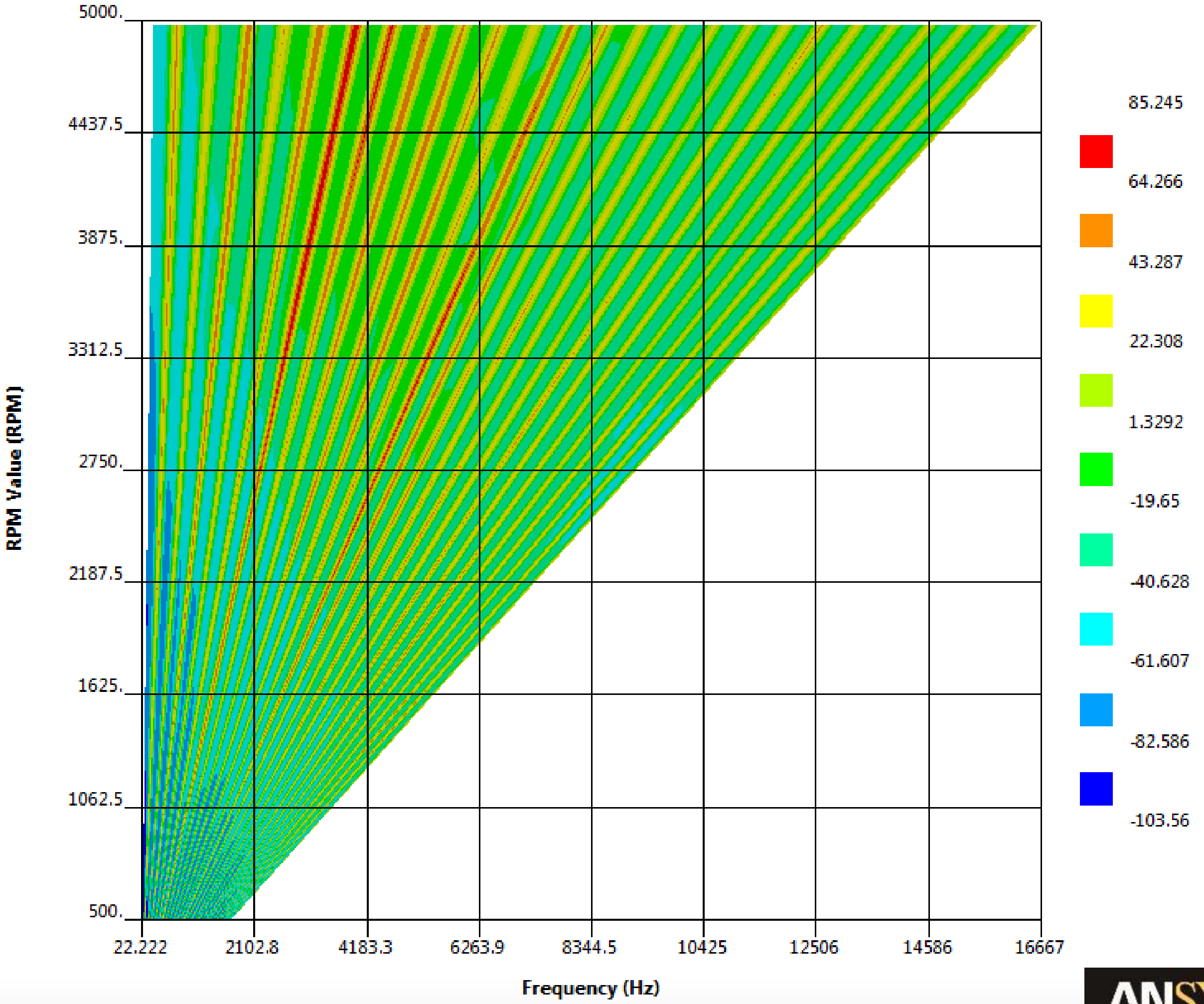




Maxwell-Mechanical: Multiple RPMs & ERP Waterfall Diagram

Equivalent Radiated Power waterfall diagram results provide a global acoustic signature for a range of RPMs and frequencies.

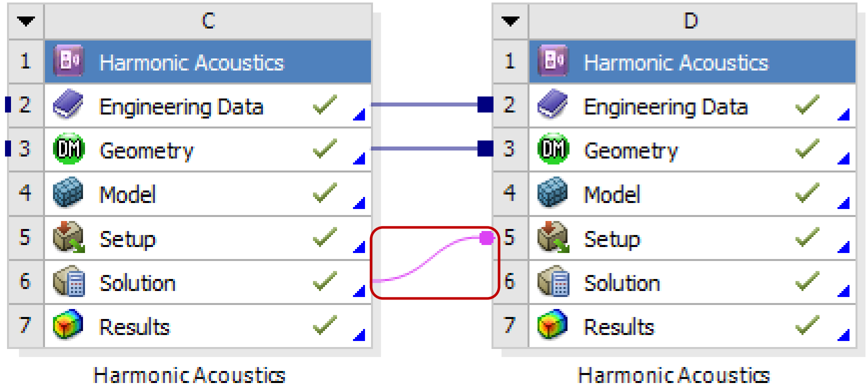


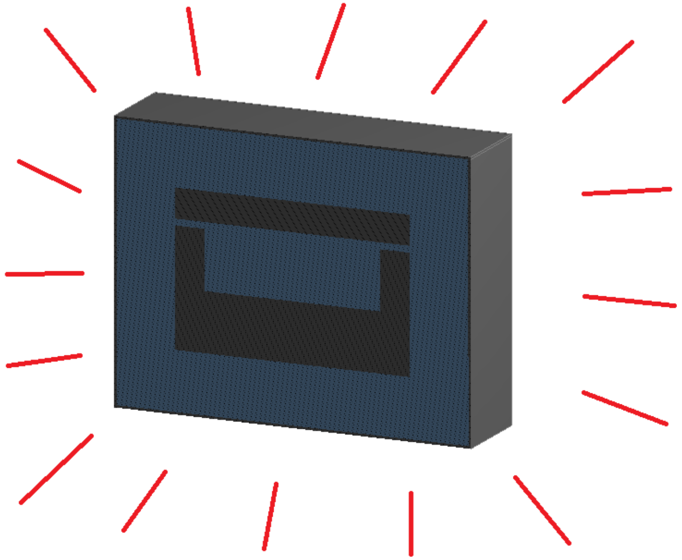


Harmonic Acoustics to Harmonic Acoustics Coupling

FSI Harmonic Acoustics can be coupled to downstream Harmonic Acoustics.

Interesting to analyse the radiated noise of a structure containing liquid (transformer …)





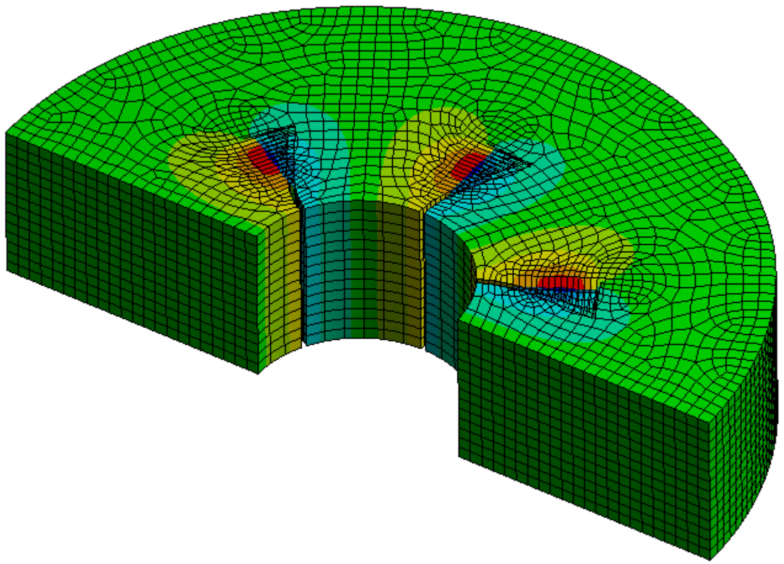
Cyclic expansion option for FSI cyclic symmetry

Cyclic Expansion options for available in Acoustics:

- ✓ Number of sectors to display
- ✓ Starting sector

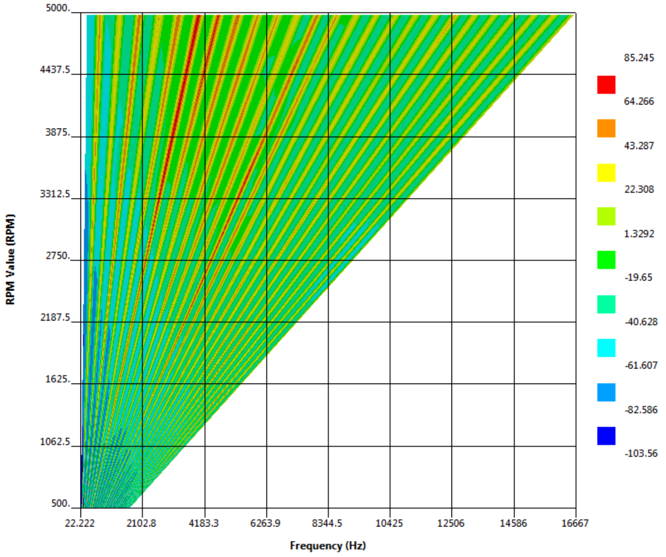
Details of "Solution (B6)"

+ Solution	
+ Adaptive Mesh Refinement	
+ Information	
− Cyclic Solution Display	
Number of Sectors	3.
Starting at Sector	1.
Expansion Method (Beta)	New

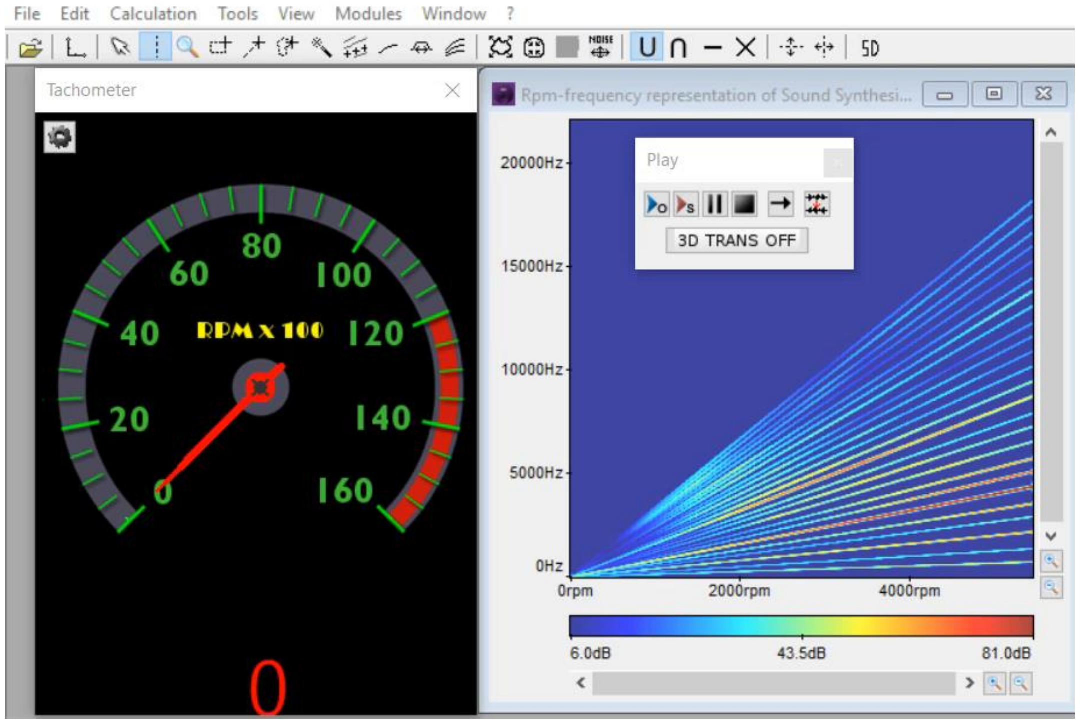


Export Results to XML format for VRXP Sound Dimension (Beta)

Allows XML export for Equivalent Radiated Power and Sound Power Level. XML file can be read in Optis VRXP Sound Dimension to synthesize the sound and create a .WAV file.



Listen to **computed** ERP



Option to ignore damping in modal Acoustics (Beta)

Available option to ignore acoustic damping material properties (viscosity…) which are defined by default for Air and Water materials. That allows to avoid using Damped solver without deleting those properties in Engineering Data.

Details of "Analysis Settings"

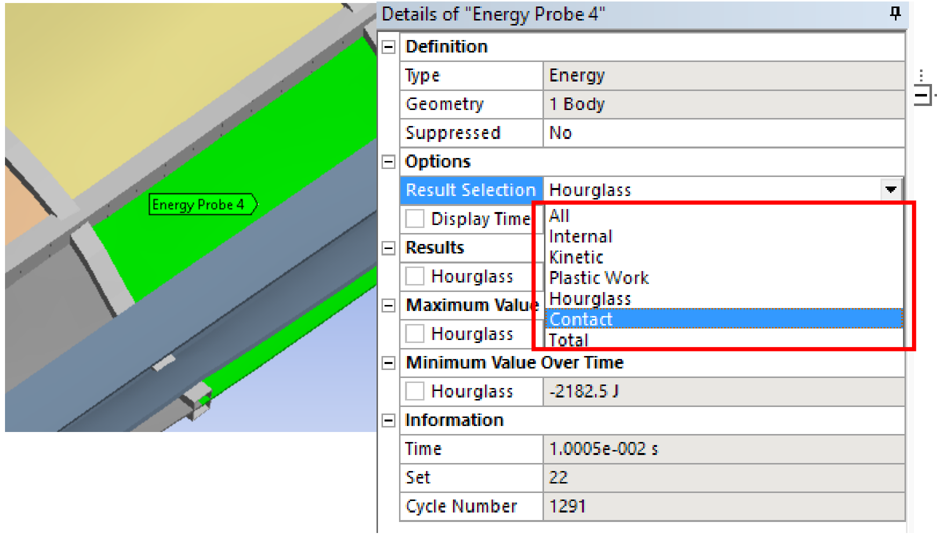
Options	
Max Modes to Find	6
Limit Search to Range	Yes
Range Minimum	1.e-002 Hz
Range Maximum	1.e+006 Hz
Solver Controls	
Damped	No
Solver Type	Program Controlled
Output Controls	
Damping Controls	
Ignore Acoustic Damping (Beta)	Yes

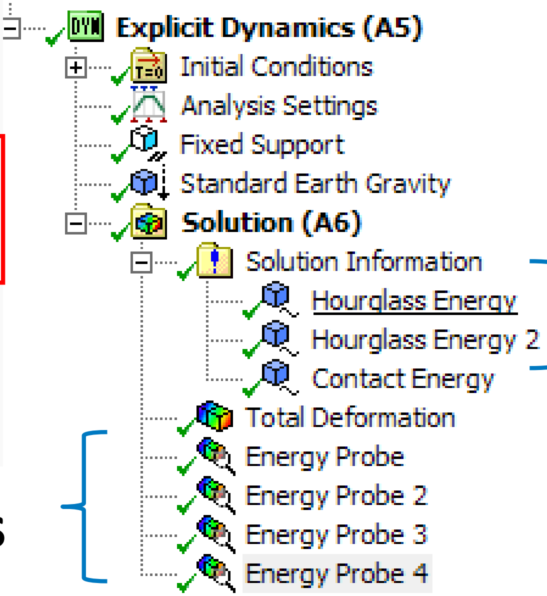
ANSYS

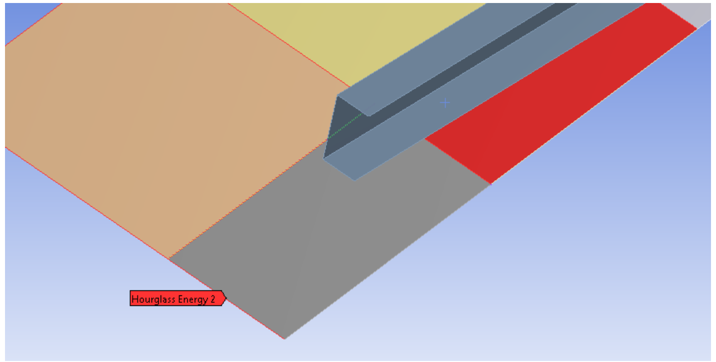
Mechanical Explicit Dynamics

ANSYS 2019 R1 update

Mechanical Explicit Dynamics – 2019 R1



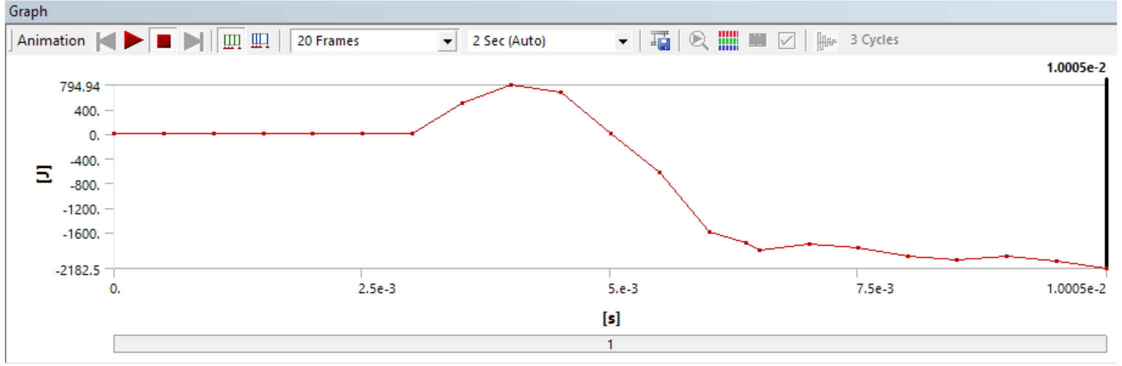


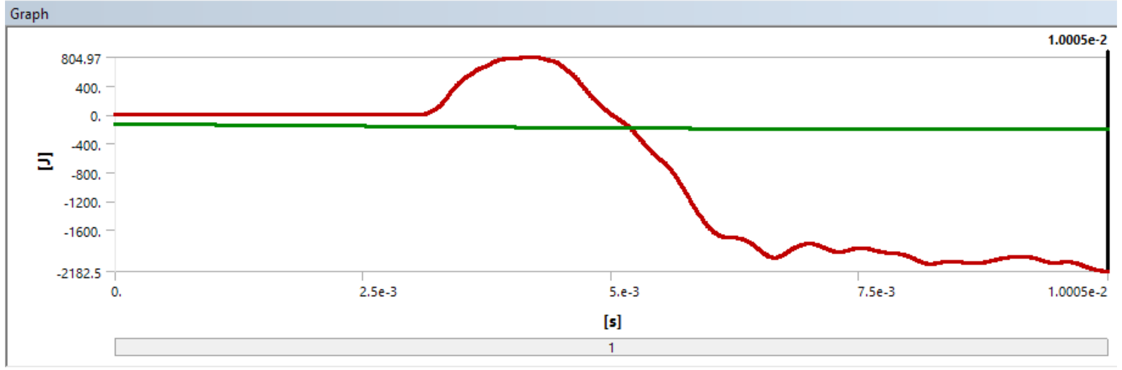


Energy Trackers now include:

- **Hourglass & Contact Energy**

Energy Probes





Energy probes can be added <u>after</u> the solve

Energy Trackers need to be defined <u>before</u> the solve

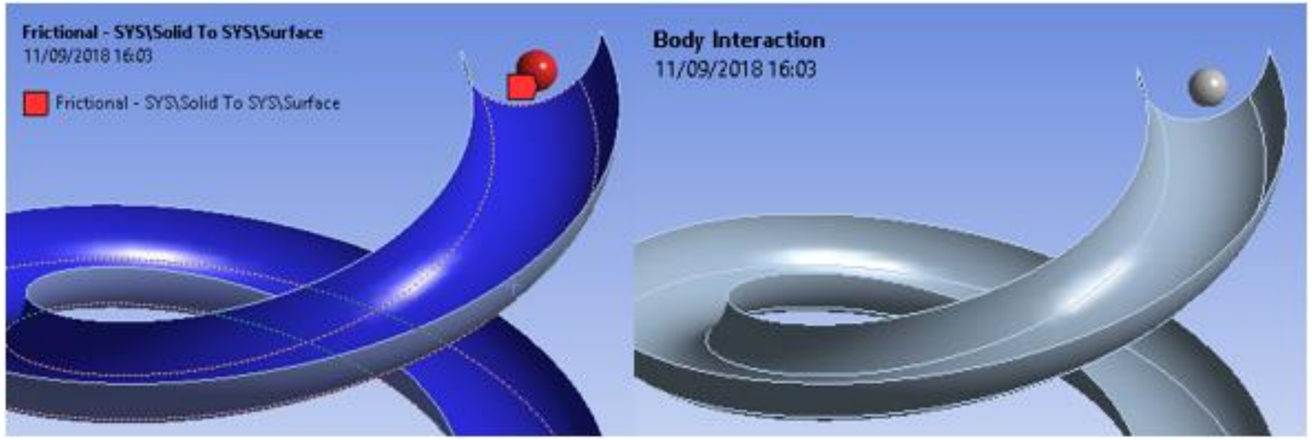
Mechanical Explicit Dynamics – 2019 R1

New Manual Contact Treatment = <Lumped/Pairwise>

- <Lumped>: default – original behaviour
- <Pairwise>: new behaviour:
  - **Affects the *Manual Contact Regions:***
    - Scoping:
      - **Contact region: nodes in contact**
      - **Target region: faces in contact**
    - Symmetry behaviour:
      - **Accounted for in the solver**
    - Friction:
      - **Definition is applied per contact pair**
      - **Should not conflict – if so: error.**

Details of "Body Interactions"

Advanced	
Contact Detection	Trajectory
Formulation	Penalty
Sliding Contact	Connected Surface
Manual Contact Treatment	Pairwise
Shell Thickness Factor	Lumped
Nodal Shell Thickness	Pairwise
Body Self Contact	Program Controlled
Element Self Contact	Program Controlled
Tolerance	0.2



Mechanical Explicit Dynamics – 2019 R1

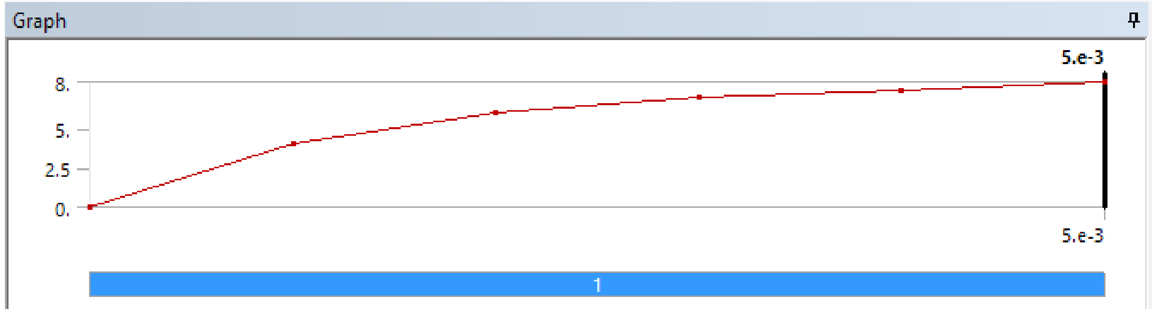
- The worksheet for stepawareness is available now for Damping Controls

Analysis Settings

Properties	Step 1	Step 2	Step 3
Step Controls			
Step End Time	1.e-003	2.e-003	3.e-003
Damping Controls			
Static Damping	0.	0.	0.

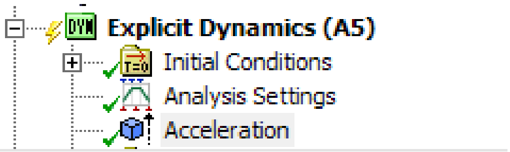
Beta

- **Tabular acceleration input**



Tabular Data

	Steps	Time [s]	Acceleration [mm/s²]
1	1	0.	0.
2	1	1.e-003	4.
3	1	2.e-003	6.
4	1	3.e-003	7.
5	1	4.e-003	7.5
6	1	5.e-003	8.
*			



Details of "Acceleration"

Scope	
Geometry	All Bodies
Definition	
Define By	Vector
Magnitude	Tabular Data
Direction	Click to Change
Suppressed	No

ANSYS

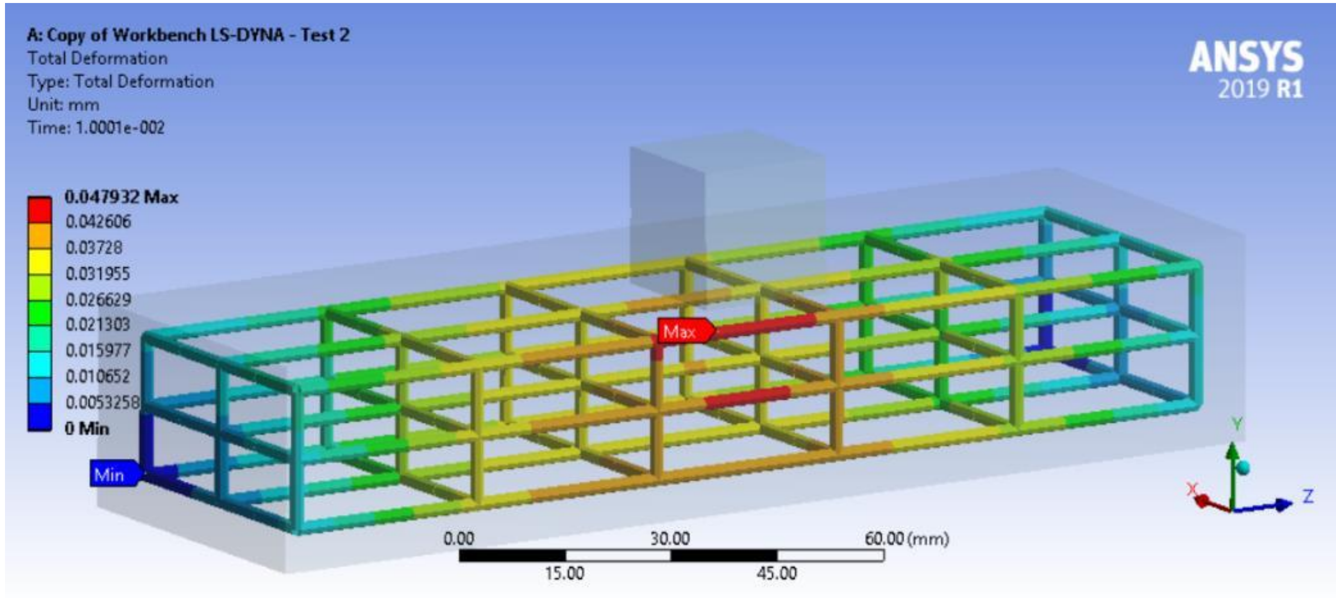
Workbench LS-DYNA

ANSYS 2019 R1 update

Body Interactions of Type Reinforcement

Body Interactions with type reinforcement are now available with LS-DYNA. They allow modeling of reinforced solid structures, like reinforced concrete.

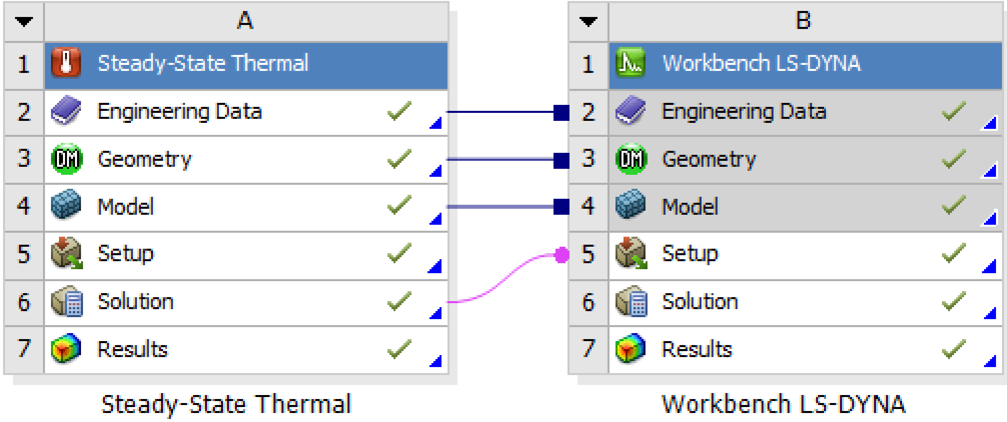
Nodes in the reinforcement beams and the matrix in which they sit do not have to be coincident.

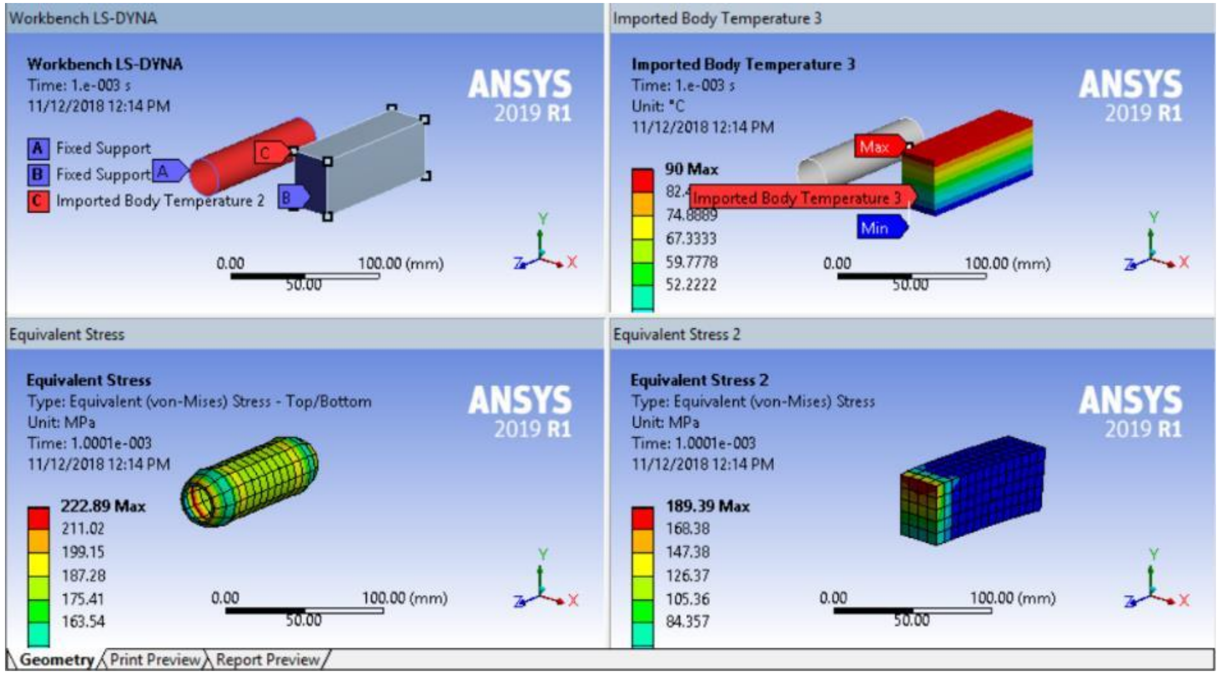


Imported Temperature

Transfer links have been enabled between steady state thermal , transient thermal calculations and Workbench LS-DYNA. allowing to transfer body temperatures from a thermal calculation to Workbench LS-DYNA.

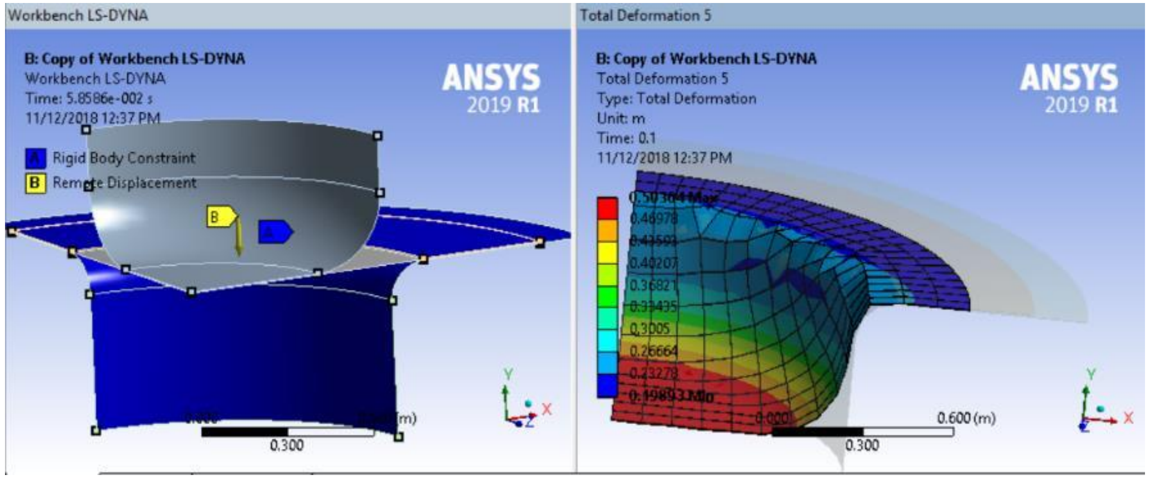
Temperature induced deformations can now be taken account in a LS-DYNA explicit calculation

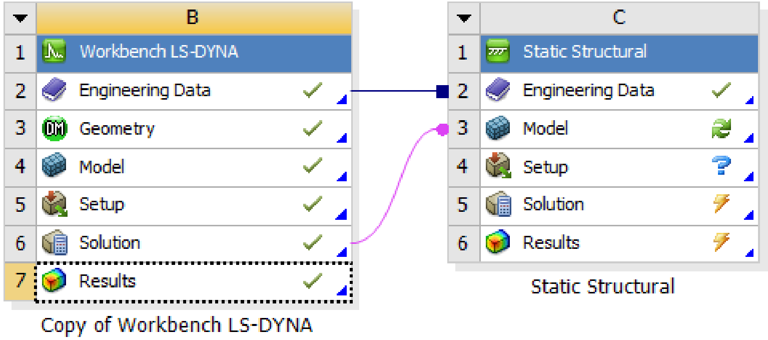


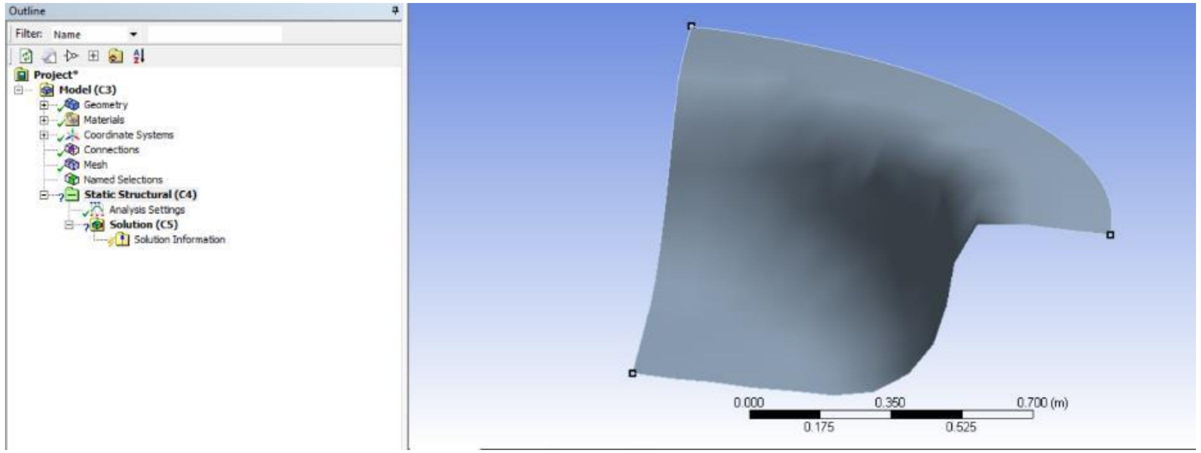


Deformation Transfer

The deformation from a Workbench LS-DYNA calculation , can now be transferred to downstream systems , like Static Structural , allowing a user to initiate those simulations from a deformed state.



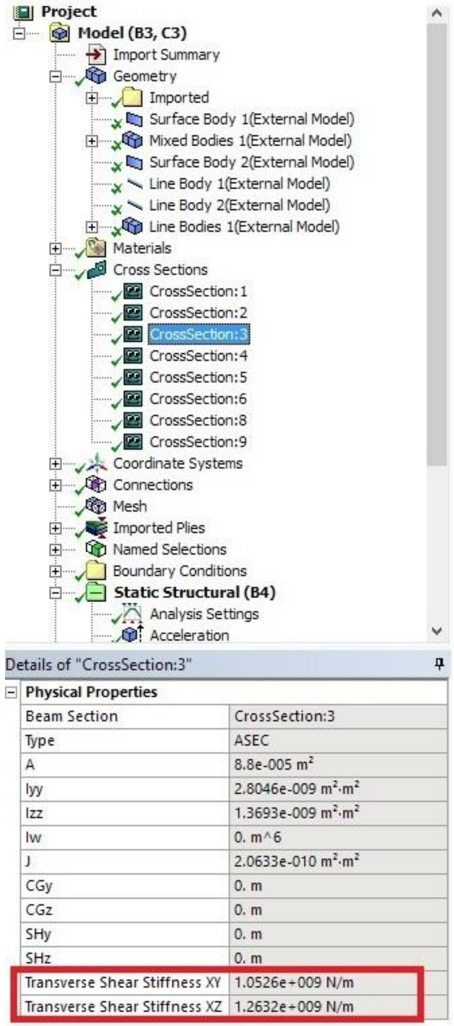




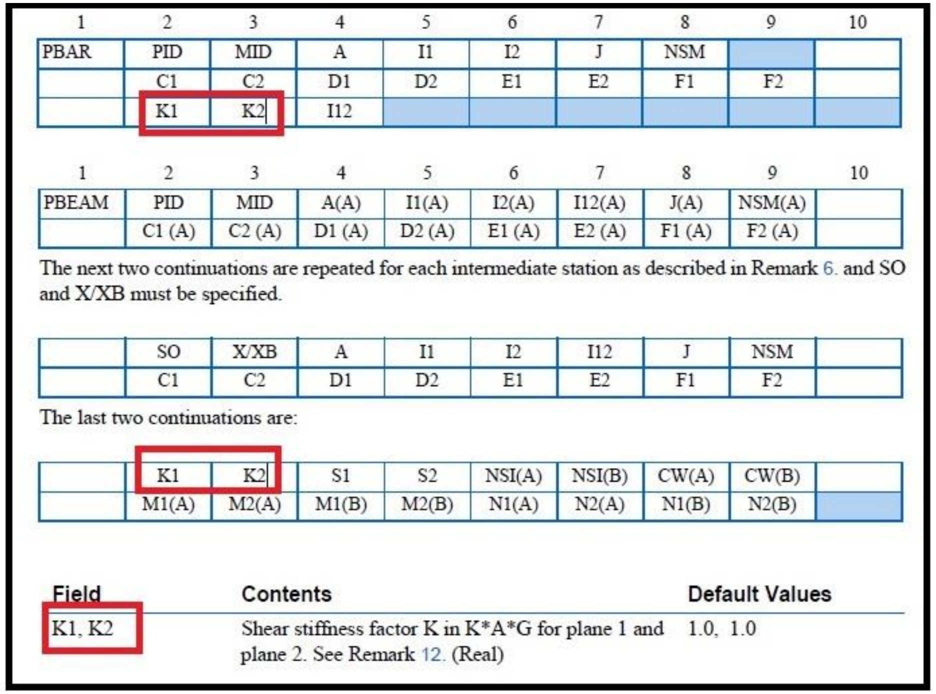
External Model

ANSYS 2019 R1 update

Import Beam shear transverse stiffness for Nastran



Beam Section	CrossSection:3
Type	ASEC
A	8.8e-005 m^2
Iyy	2.8046e-009 $m^2 \cdot m^2$
Izz	1.3693e-009 $m^2 \cdot m^2$
Iw	0. m^6
J	2.0633e-010 $m^2 \cdot m^2$
CGy	0. m
CGz	0. m
SHy	0. m
SHz	0. m
Transverse Shear Stiffness XY	1.0526e+009 N/m
Transverse Shear Stiffness XZ	1.2632e+009 N/m



1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A(A)	I1(A)	I2(A)	I12(A)	J(A)	NSM(A)	
	C1 (A)	C2 (A)	D1 (A)	D2 (A)	E1 (A)	E2 (A)	F1 (A)	F2 (A)	

The next two continuations are repeated for each intermediate station as described in Remark 6. and SO and X/XB must be specified.

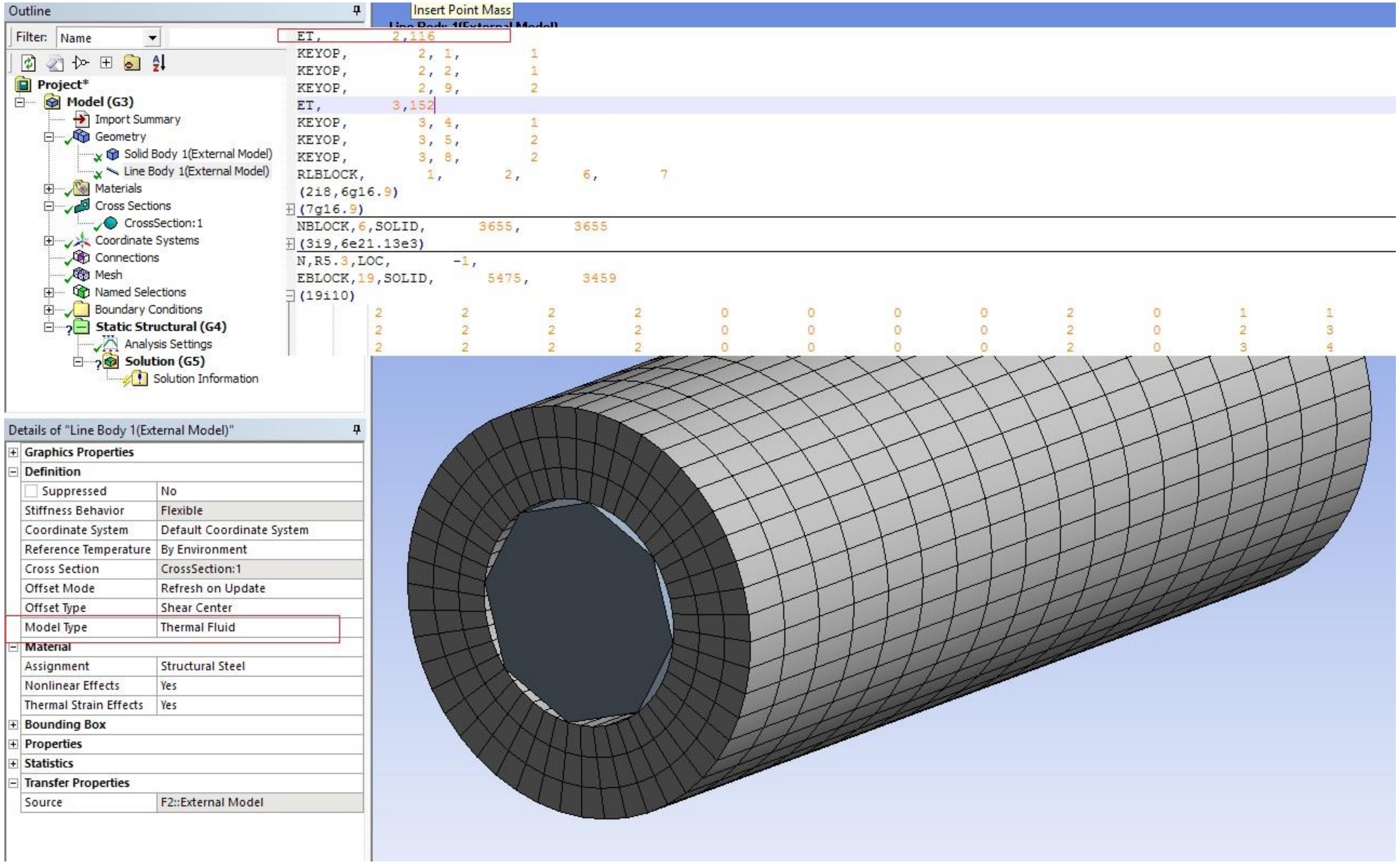
	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	D1	D2	E1	E2	F1	F2	

The last two continuations are:

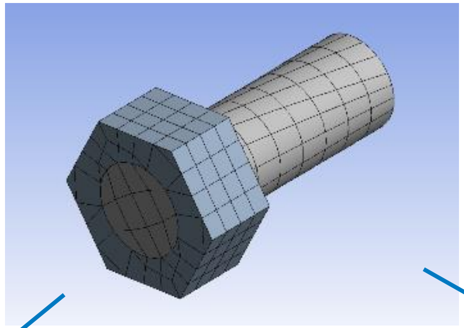
	K1	K2	S1	S2	NSI(A)	NSI(B)	CW(A)	CW(B)	
	M1(A)	M2(A)	M1(B)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	

Field	Contents	Default Values
K1, K2	Shear stiffness factor K in K*A*G for plane 1 and plane 2. See Remark 12. (Real)	1.0, 1.0

Import fluid116 elements for MAPDL

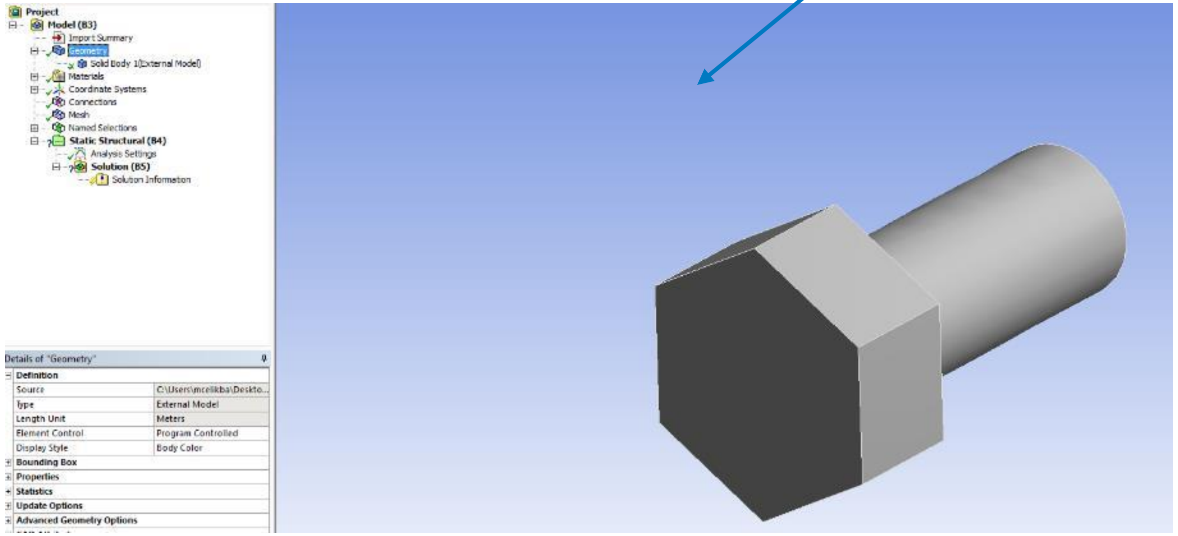


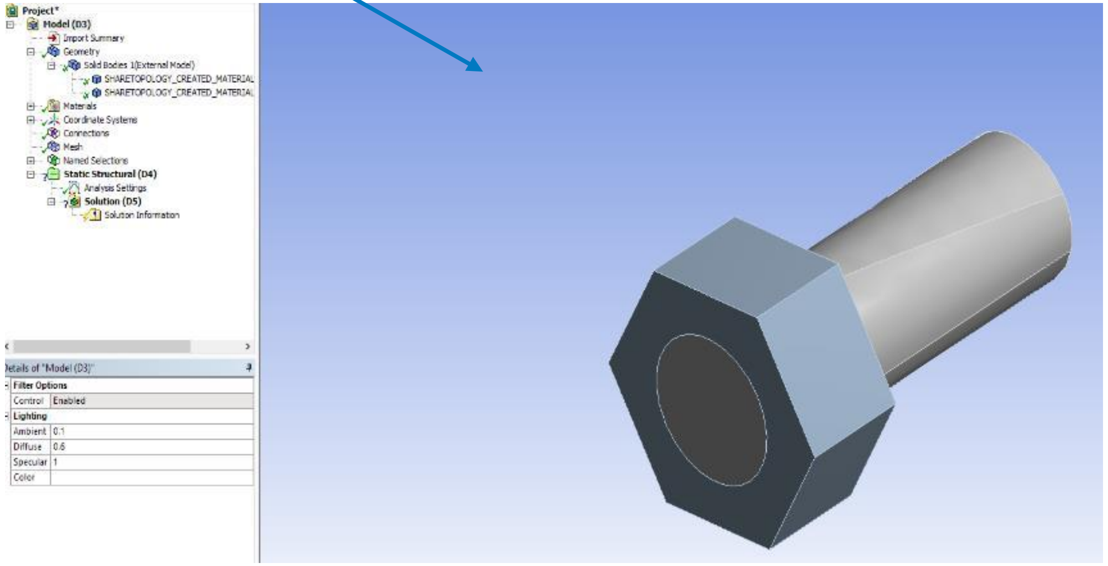
Import ICEM .uns files with topology (Bodies/Faces)



v192

2019R1



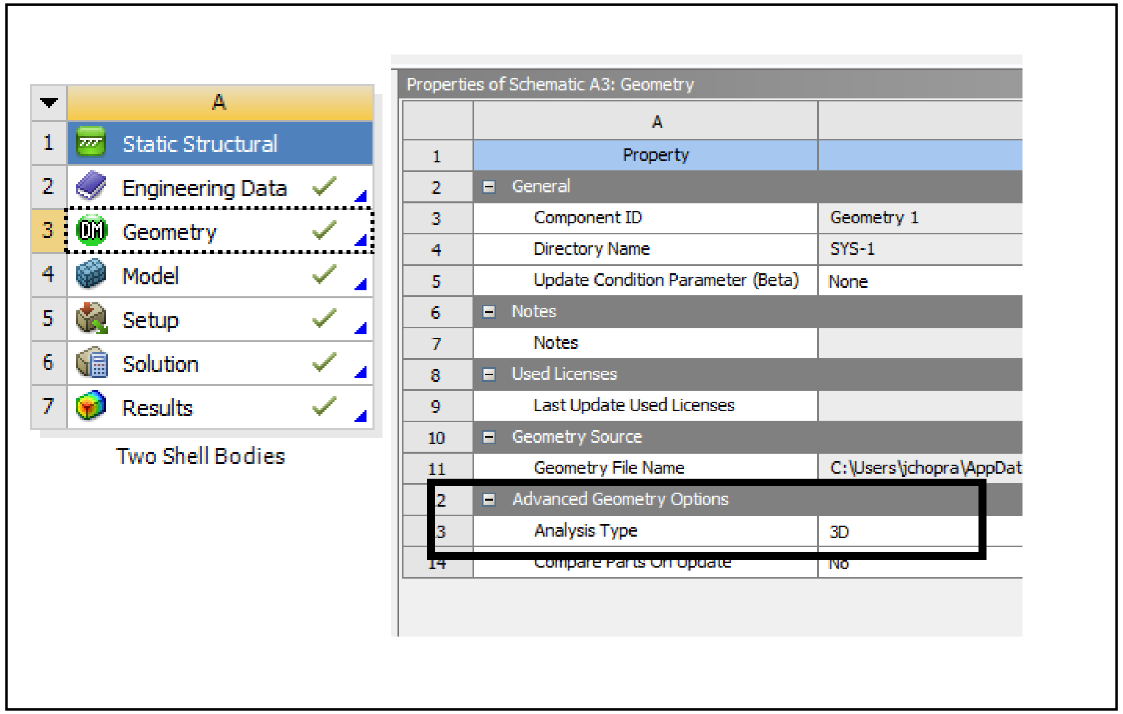


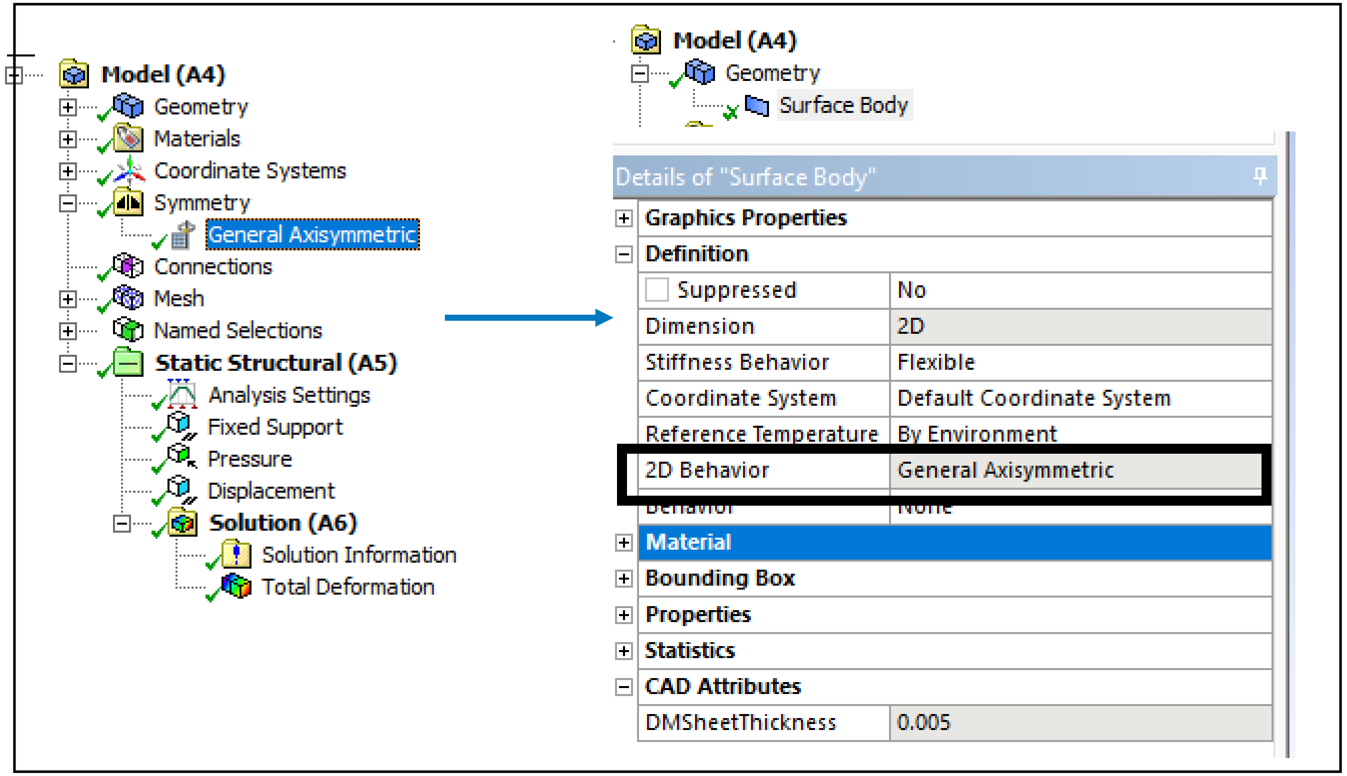
2019 R1 General Axisymmetric

ANSYS 2019R1 update

General Axisymmetric in WB-Mechanical

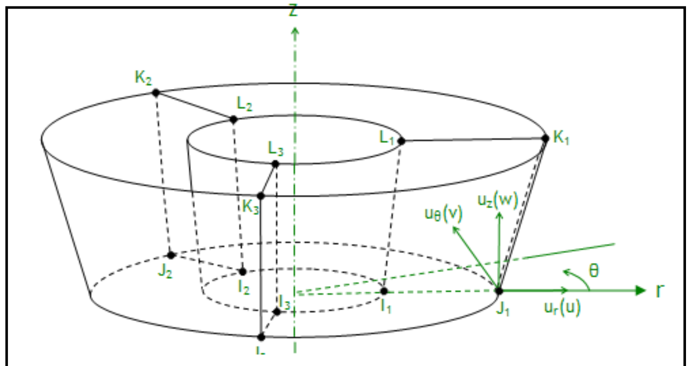
Applicable for 3D Static Structural analysis. General Axisymmetric definition is added under Symmetry folder and when scoped to 2D/Surface body makes the behavior of that Body as General Axisymmetric



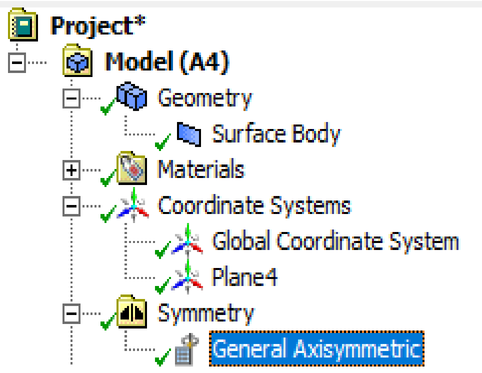


General Axisymmetric Definition

1. General Axisymmetric (Solid 272/273) introduces Fourier series into interpolation function to describe the change of displacements in the Circumferential (θ) direction.
2. General Axisymmetric object in mechanical takes Nodal planes input to define number of planes and Coordinate system with Axis input to specify the circumferential direction.
3. The graphics view shows the Axis and Orientation by drawing the line and circle. And it shows the number of nodal planes by showing the transformed geometry in different nodal planes

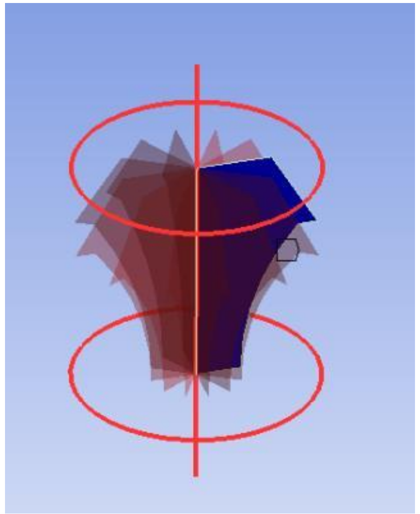


Solid 272/273



Details of "General Axisymmetric"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Nodal Planes	12
Coordinate System	Plane4
Axis	Y Axis
Suppressed	No

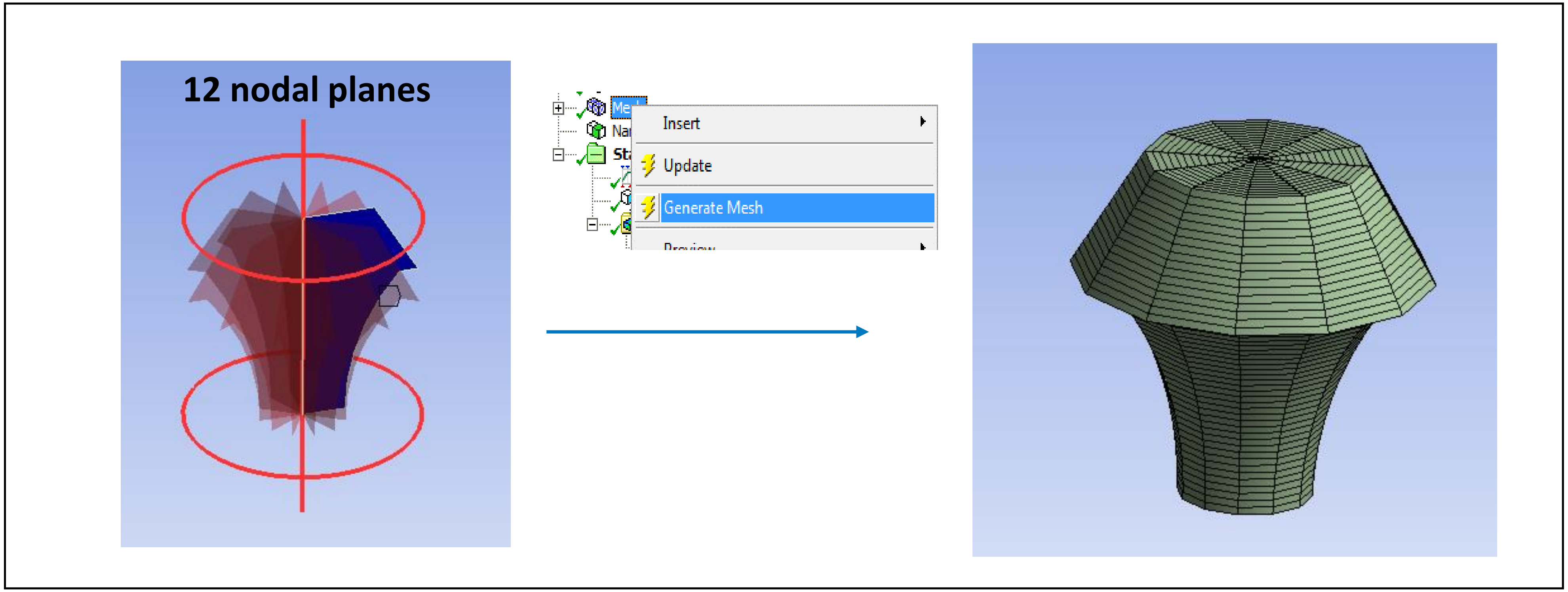


Nodal planes value range from 1 to 12 excluding 2

Axis definition must not cut through the body

General Axisymmetric Mesh

1. General Axisymmetric mesh is generated using Generate Mesh/Update action on Mesh folder
2. The base mesh is created on the surface body and then General Axisymmetric mesh is generated on all the nodal planes as post operation on base mesh.



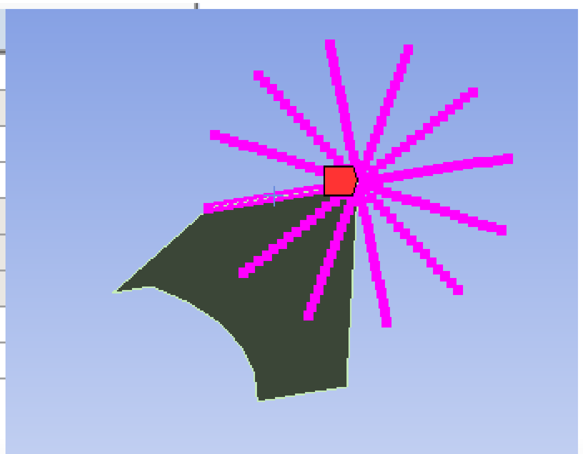
Loads and Boundary Conditions

1. Nodal loads are directly applied through the Named selections scoped to Nodes. These loads can be non-axisymmetric loads as nodes can be picked from any nodal plane
2. Pressure, Remote force, Moment and Displacement load can be applied to the geometric scoping which can be edge or vertex. If edge of General Axisymmetric body is selected for load application, then load is applied to all the nodal planes. For Pressure load using Surface effect option, SURF159 is created to apply the load

Load applied to nodes

Details of "Nodal Pressure"

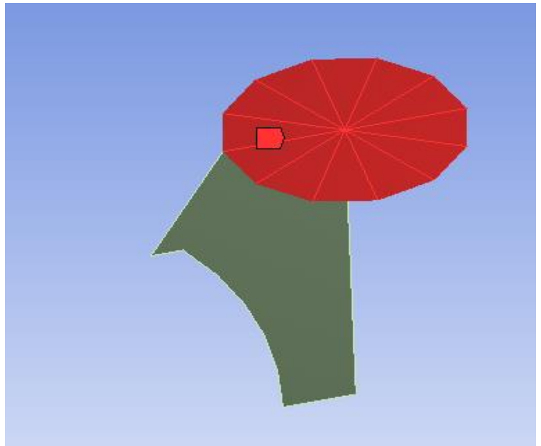
Scope	
Scoping Method	Named Selection
Named Selection	Selection 2
Definition	
ID (Beta)	232
Type	Pressure
Define By	Normal To
Magnitude	-1.e+006 Pa (ramped)
Suppressed	No



Graphics shows the face selected in θ direction where load is applied when Pressure is scoped to edge

Details of "Pressure"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
ID (Beta)	212
Type	Pressure
Define By	Normal To
Applied By	Surface Effect
Magnitude	-1.e+006 Pa (ramped)
Suppressed	No

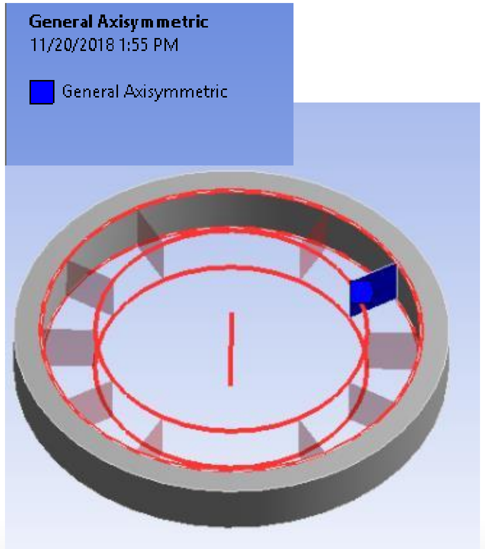


General Axisymmetric behavior with Contacts

1. Node to surface contact is created between General Axisymmetric body in contact with other bodies. CONTA175 is created for the contact side which will be General Axisymmetric body and TARGET170 element is created for 3D target surface

2. Only Bonded contact is supported when Nodal plane 1 is defined

3. Number of nodal plane should be same when General Axisymmetric body is in contact with other General Axisymmetric body

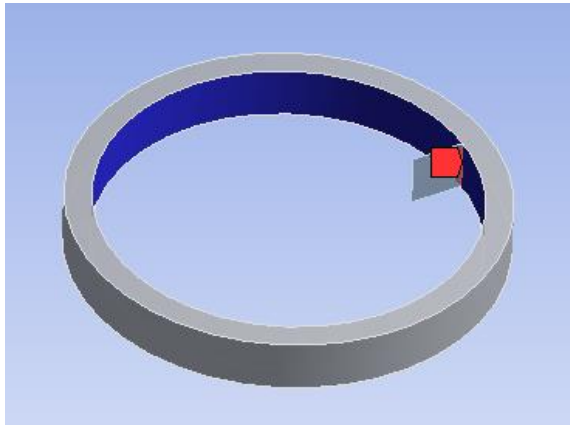


Edge to Face contact between General Axisymmetric body and Solid body

Details of "Bonded - Inner_Ring_2D To Outer_Ring"

Scope	
Scoping Method	Geometry Selection
Contact	1 Edge
Target	1 Face
Contact Bodies	Inner_Ring_2D
Target Bodies	Outer_Ring
Protected	No
Definition	
Type	Bonded
Scope Mode	Manual
Suppressed	No
Advanced	
Formulation	Program Controlled
Small Sliding	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled
Geometric Modification	
Target Geometry Correction	None

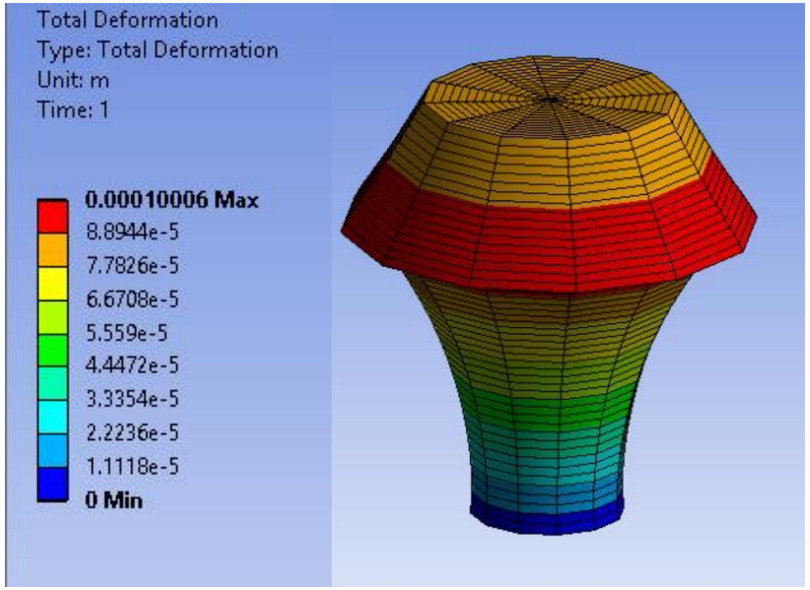
Nodes in all nodal planes for the General Axisymmetric scoped edge is considered for contact



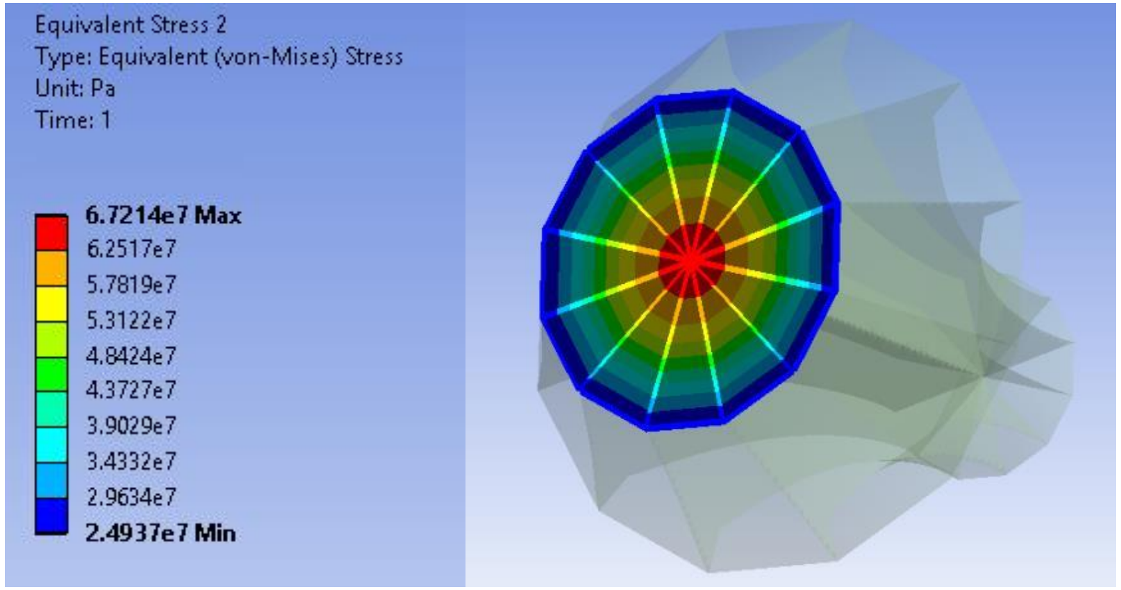
Results

1. All regular results can be extracted on the General Axisymmetric body scoping or Mesh. The results shown below are Deformation and Stress which are symmetric in the circumferential direction in the presence of Axisymmetric loading applied in this case

Total deformation scoped to All Bodies



Stress shown in all nodal planes when scoped to an edge



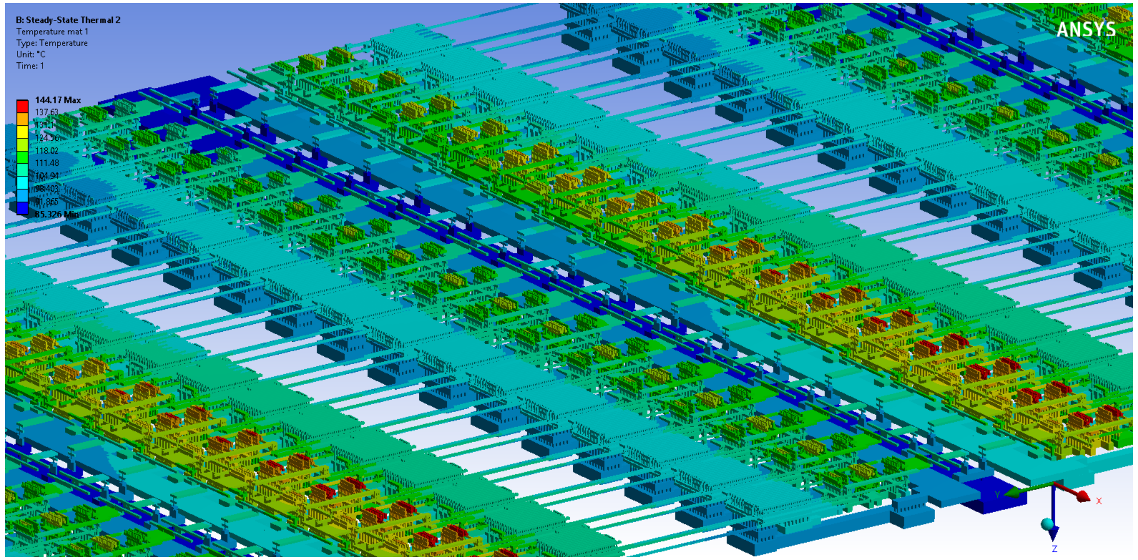
MAPDL Elements

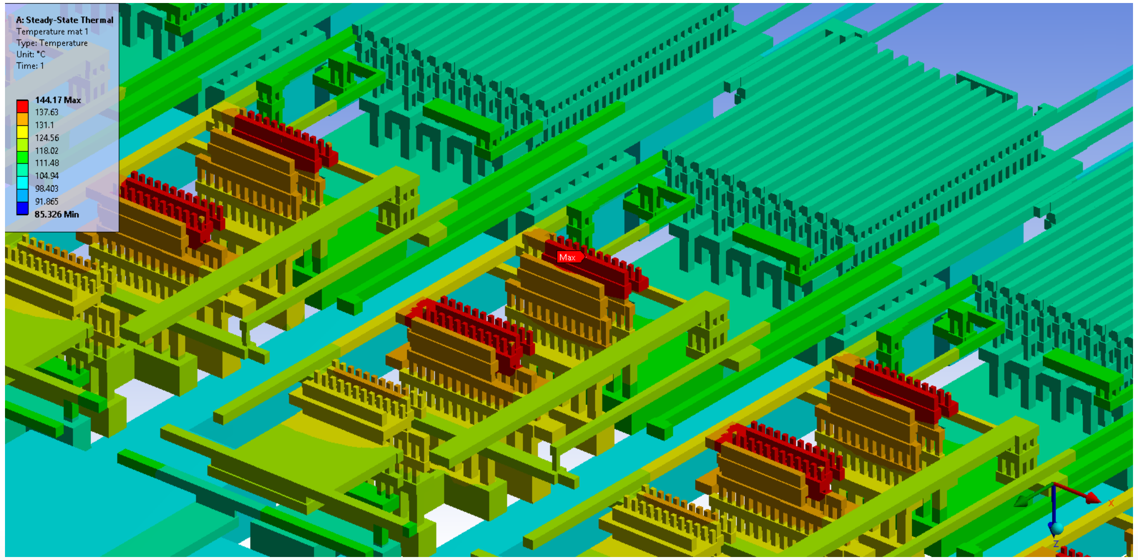
ANSYS 2019 R1 update

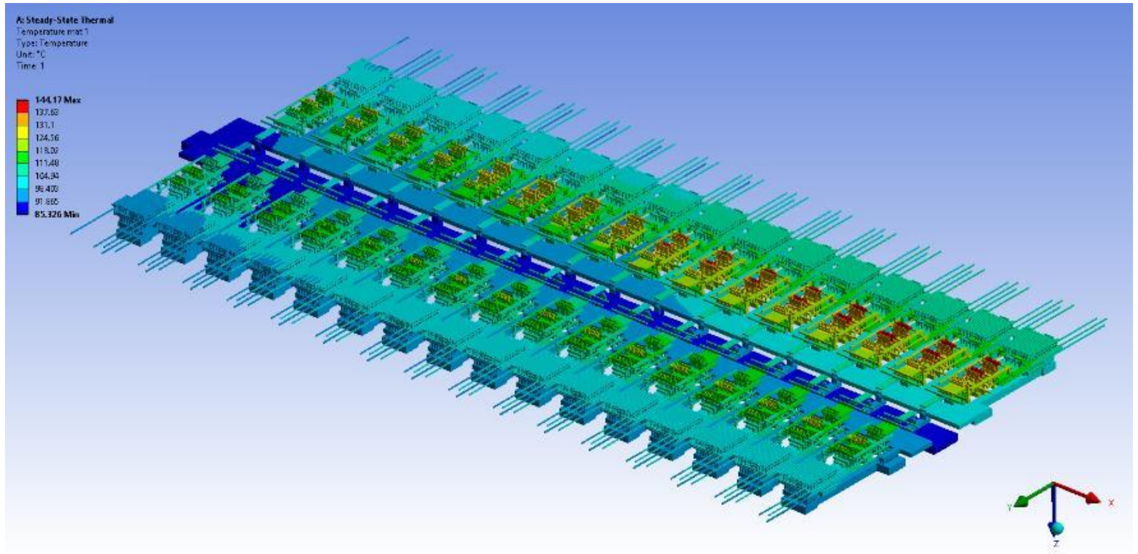
List of 2019 R1 Features

- Thermal Reinforcing Elements
- 10-Node Tetrahedral Thermal Solid
- Enhanced Pre- & Post-processing for Reinforcing Elements
- General Distributed Load for Solid Elements
- Hybrid Cable Element
- Linear Perturbation for General Axisymmetric Elements
- Anisotropic Structural and Dielectric Losses for Piezoelectric Analysis

THERMAL REINFORCING: Motivation





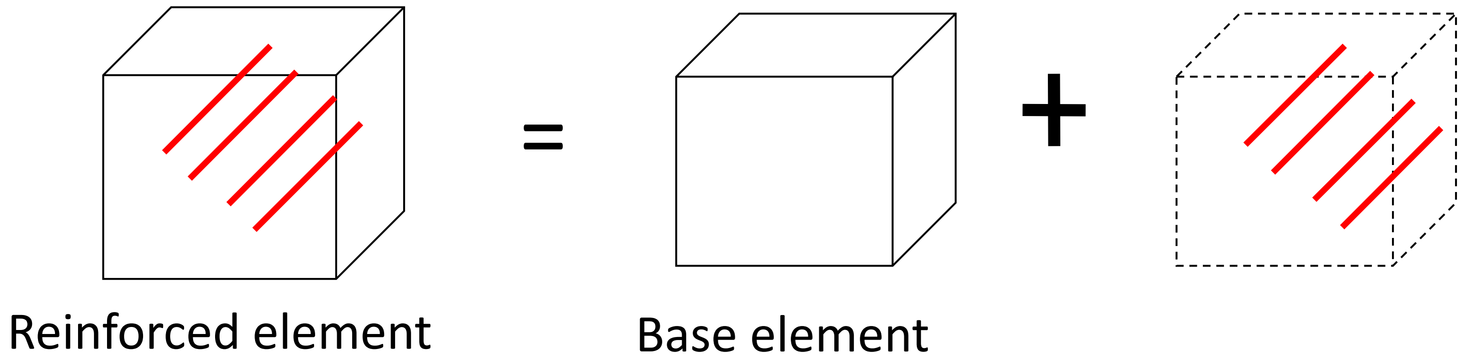


Complex PCB thermal model

~ 72 million dof with 3D detailed mesh
Extended model preparation time
Excessive computational cost

THERMAL REINFORCING: Basic Approach

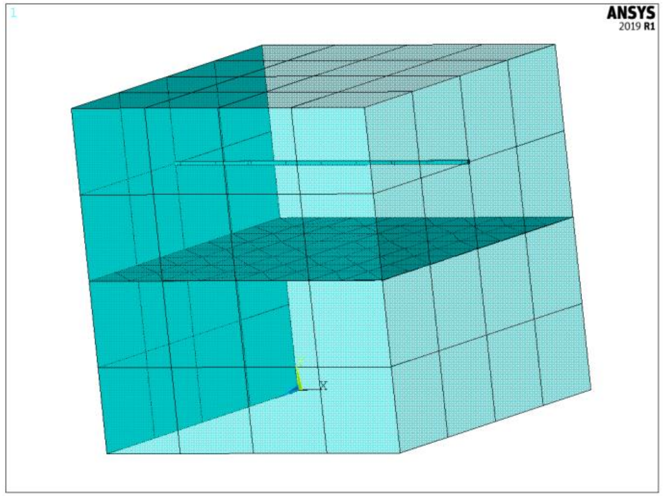
Use reduced order model for the embedded metal regions – line / plate elements
Use Reinf Technology to capture the thermal solution by embedding the line/plate Elements in a base element



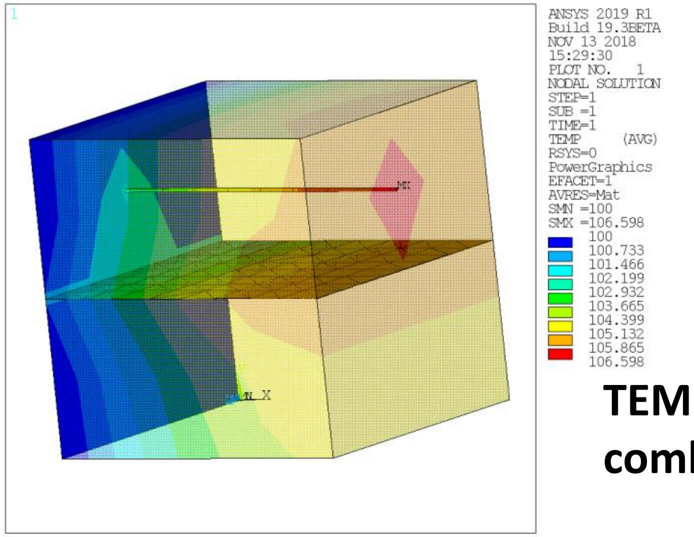
THERMAL REINFORCING: 2019 R1 Scope

- 3D Discrete Reinforcing element REINF264 for SOLID278/279
- 3D Smeared Reinforcing element REINF265 for SOLID278/279
- Support both uniaxial and homogeneous options for Reinf265
- Provide lower & higher order options
- Allow base material removal
- Analysis options (static & transient & quasi)
- The base elements must be homogeneous (ie. KYOP3=0)
- EREINF command modification for thermal to structural conversion and load application

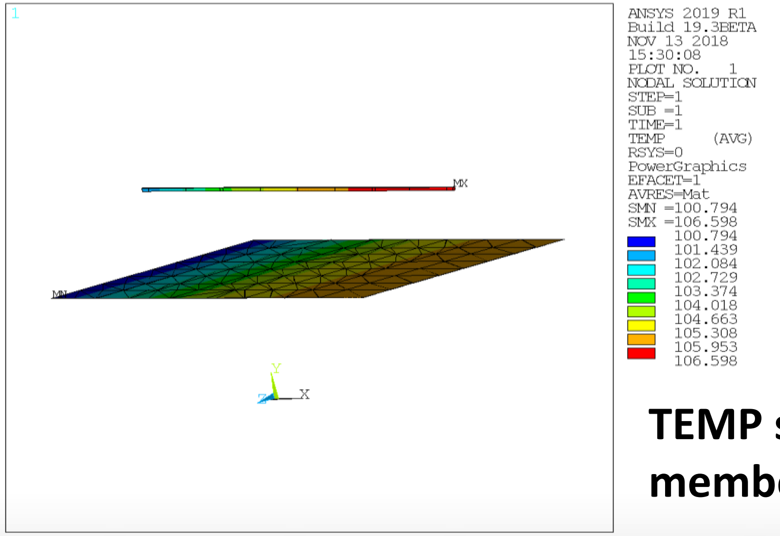
THERMAL REINFORCING: Example



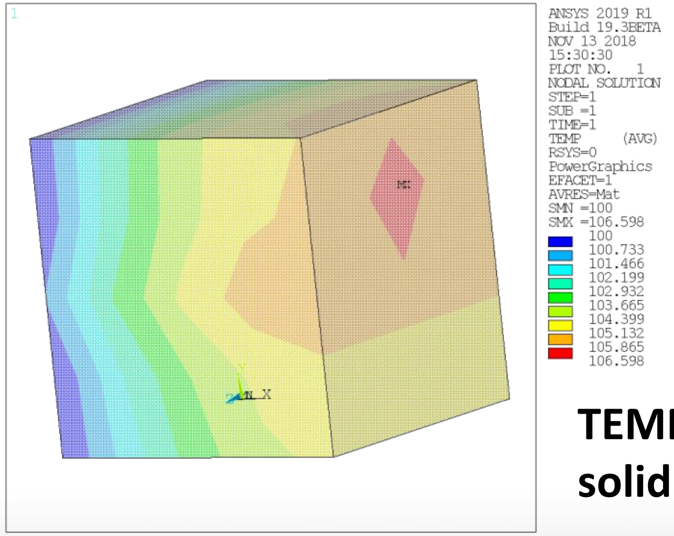
- **Discrete and smeared members**
- **Heat generation load on the members**



TEMP solution on the combine model



TEMP solution on the members



TEMP solution on the base solid elements

Command Enhancements for Reinforcing Elements

EGID – new command to specify group Id for mesh200 element

EMODIF – allow EGID modification for all elements

EREINF – allow for REINF element update based on base element

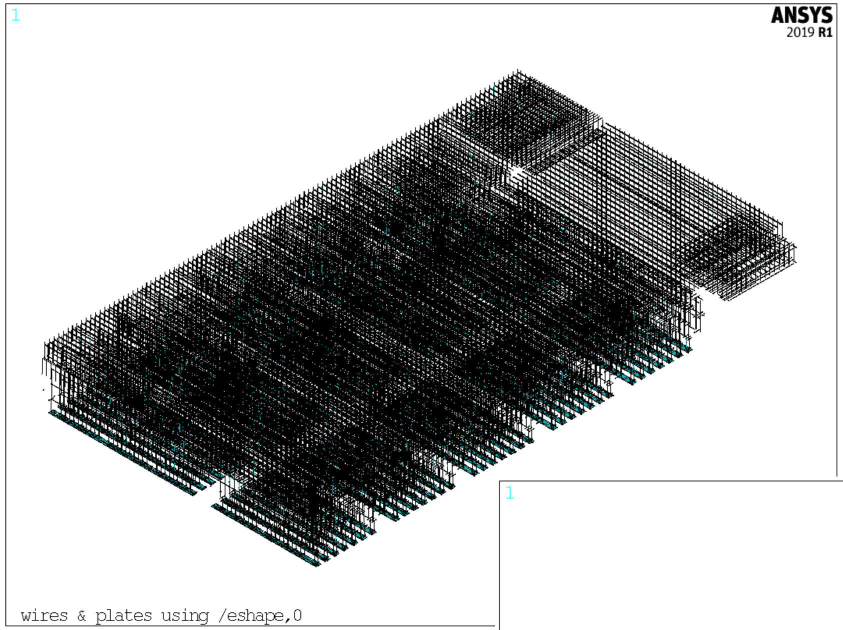
BFE – can be used with mesh200, reinf264, reinf265 for HGEN

*GET – modified to return group identifier, number of members for selected REINF element

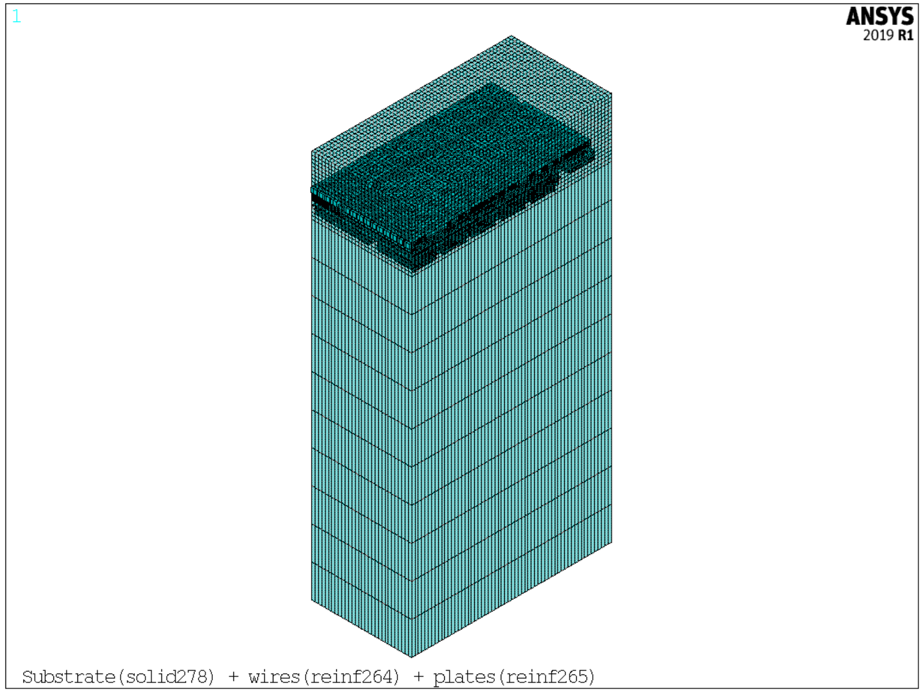
*GET – modified to return min/max TEMP for specified or selected group identifier

*VGET – modified to return array of min/max TEMPS for specified or selected group identifier

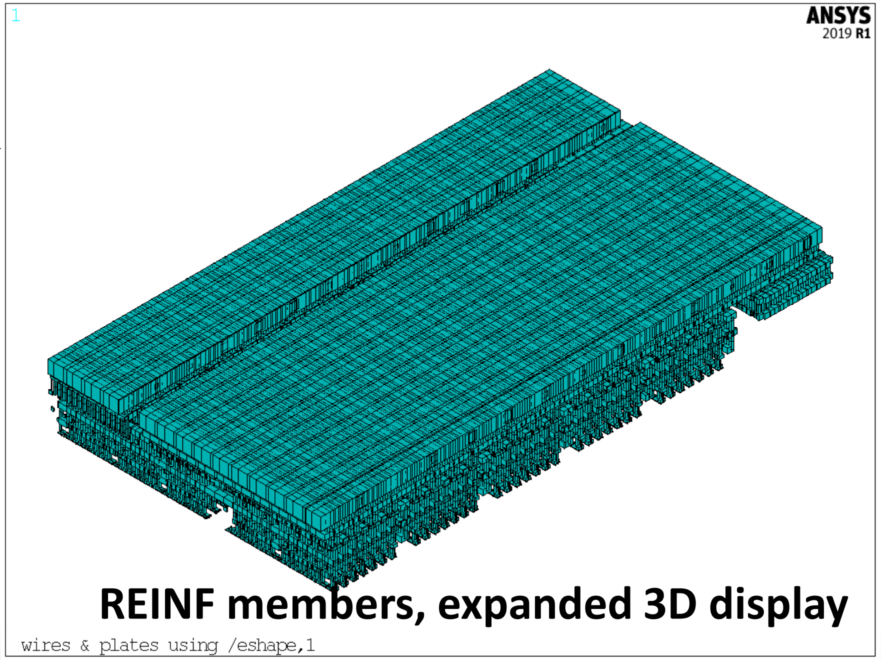
THERMAL REINFORCING: Chip Thermal Analysis



REINF members (/ESHAP, 0)

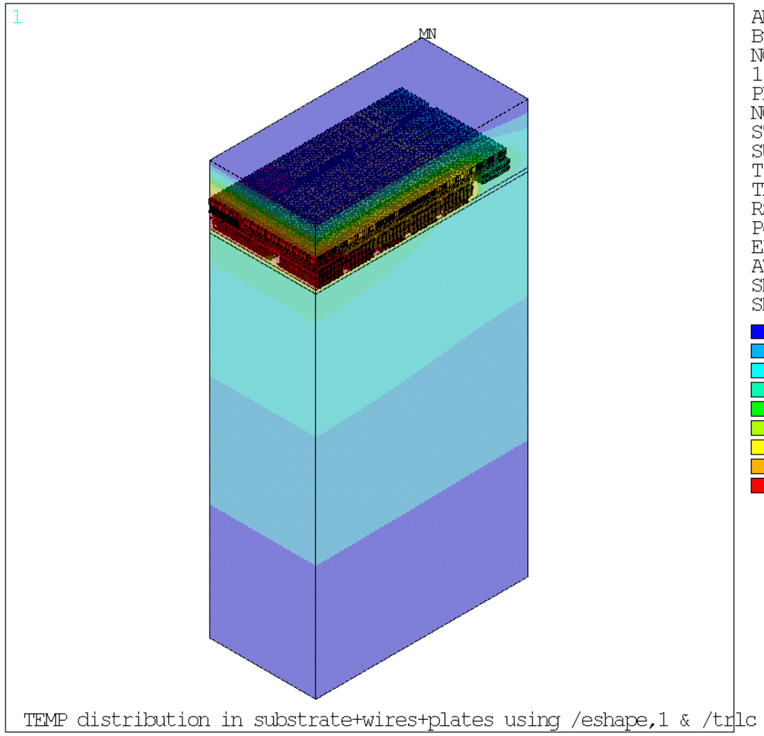


REINF model: Homogenous Substrate & Embedded Traces (Discrete and Smeared)

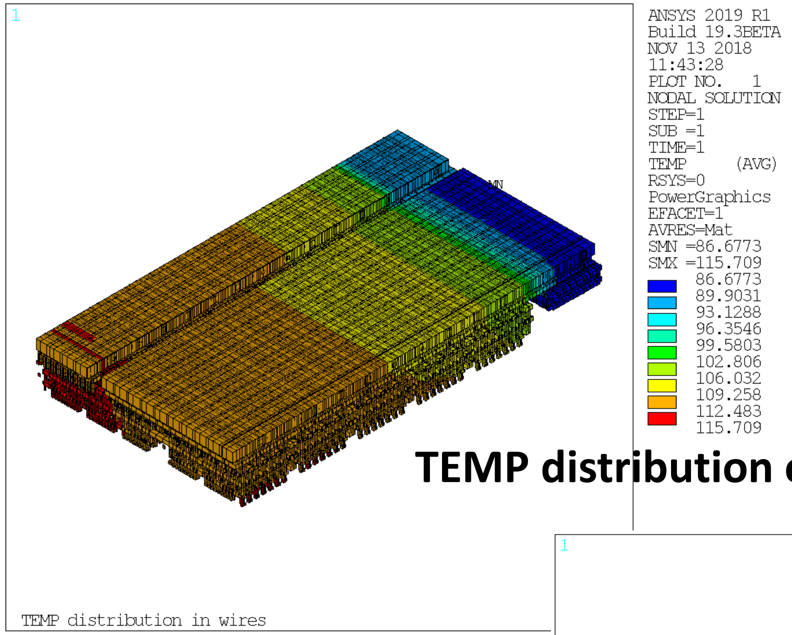


REINF members, expanded 3D display

THERMAL REINFORCING: Chip Thermal Analysis

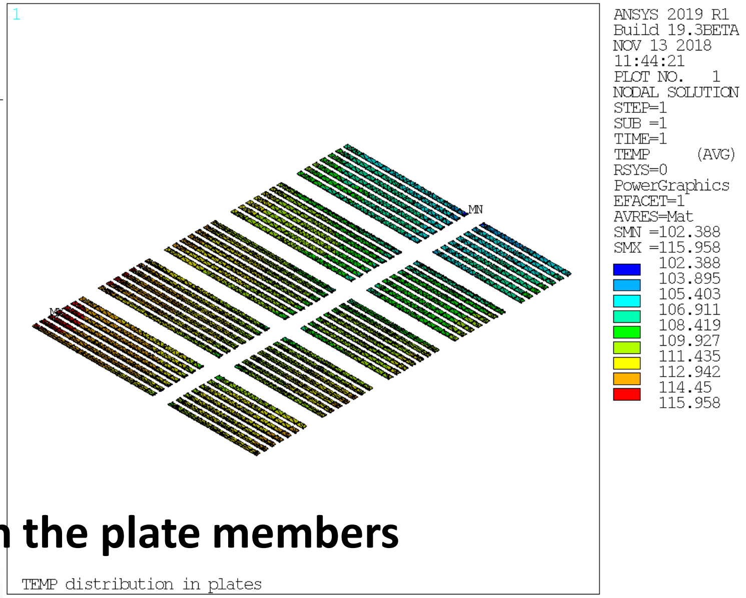


TEMP distribution on the combined model



TEMP distribution on the wire members

Members can be grouped with global identifiers (GID) and selected by GIDs for post-processing



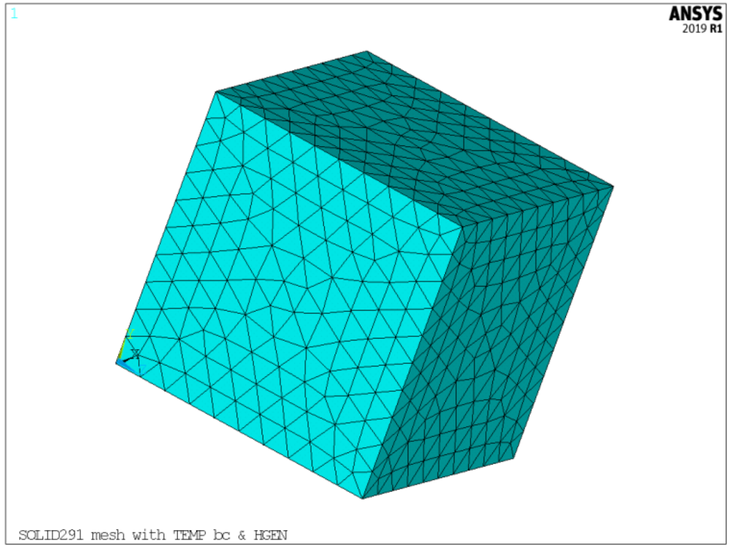
TEMP distribution on the plate members

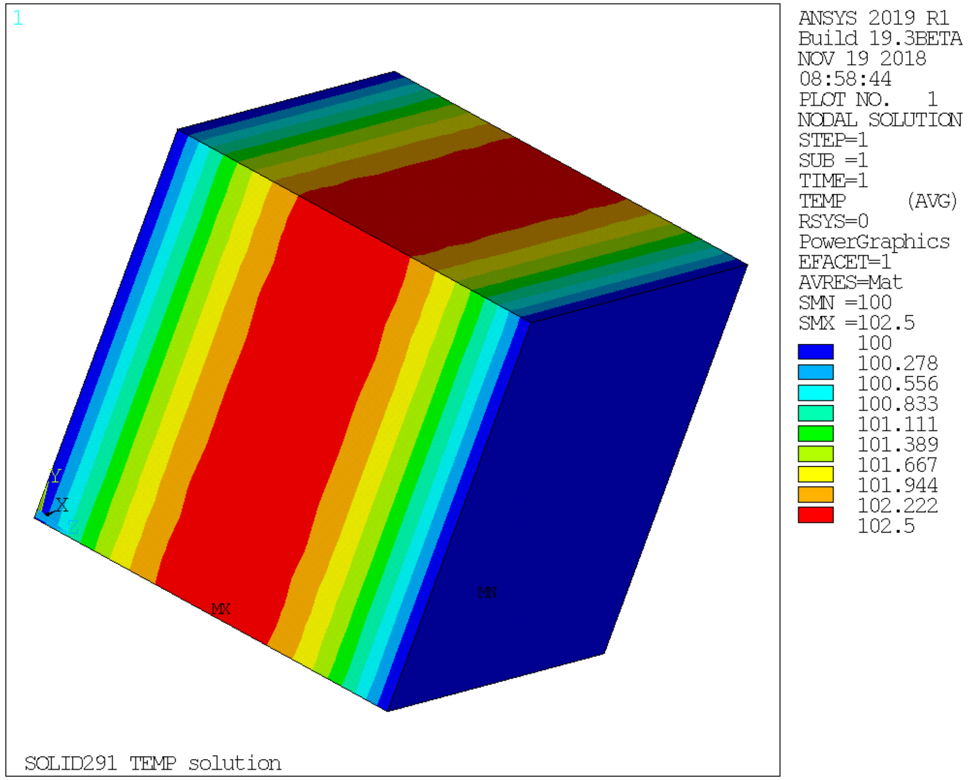
Other Thermal Enhancements: 3D 10-Node Tetrahedral

New generation 10-noded thermal tetrahedral element (beta)

-USERMATTH is supported

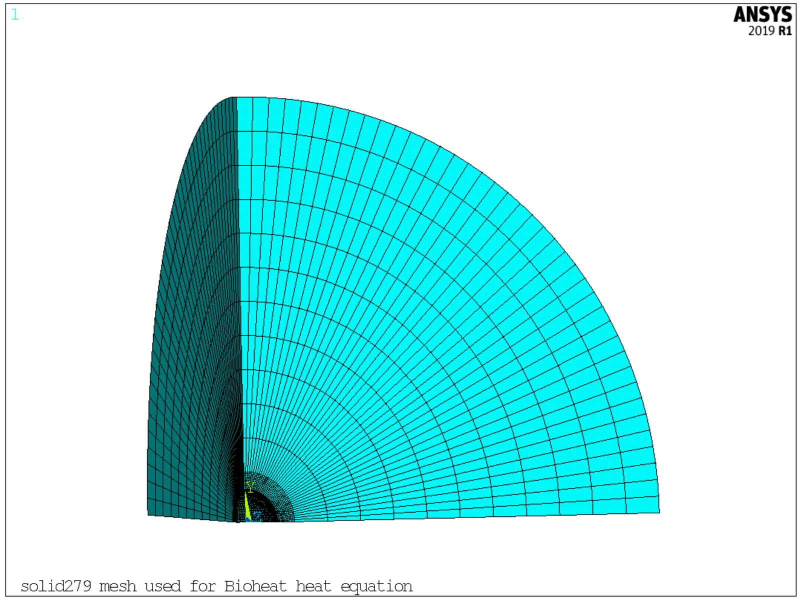
-TB commands can be used

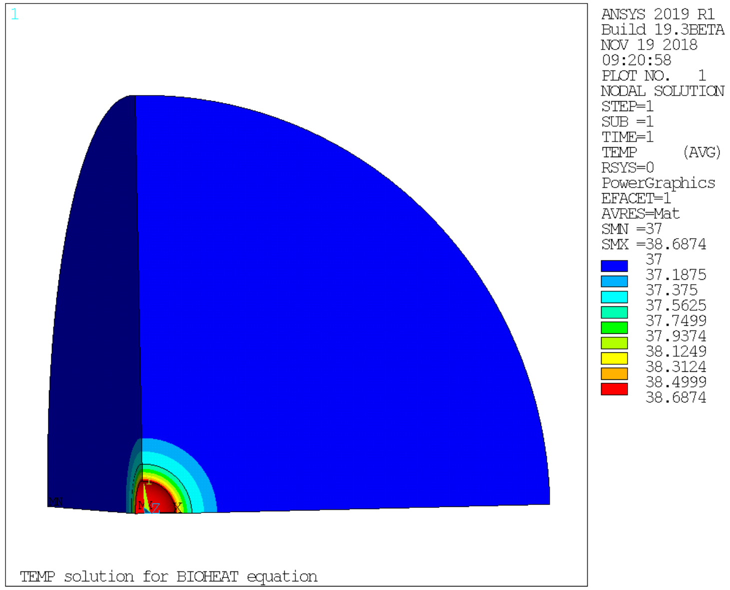




Other Thermal Enhancements: Nonlinear Heat Generation

- Overcome convergence difficulty with strongly nonlinear heat generations
- SOLID278/279 have been enhanced with consistent linearization in REV 2019 R1 to address this issue





Enhanced pre-/post-processing for Reinforcing elements

Global identifiers (GID) for reinforcing members

- EGID: Assign a Global ID to selected elements
- EMSEL: Select reinforcing members by GIDs
- Supported for visualization and result post-processing

Heat generation load on reinforcing members

- Direct application to the members
- Application to MESH200 elements in a mesh independent procedure

*GET and *VGET new capabilities

- Retrieve member information : number of members, min/max GIDs, and more
- Retrieve min/max temperature results

Support node-based Initial State via mesh independent method

Enhanced PRNSOL command for thermal reinforcing result listing

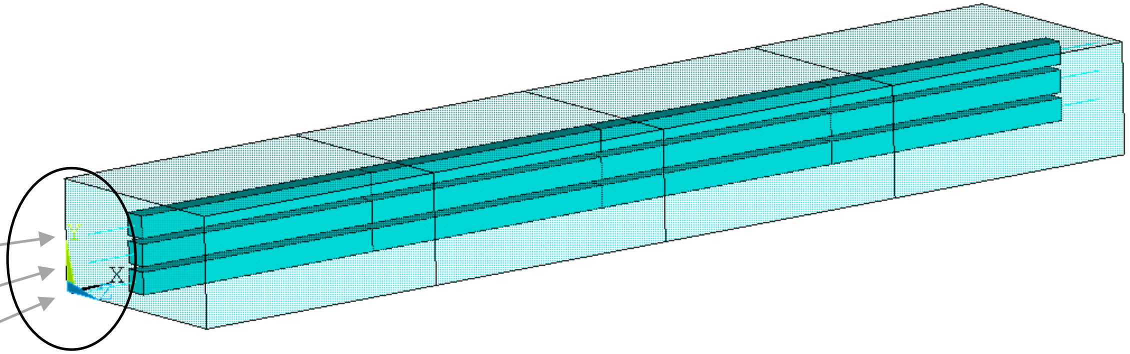
Application of GIDs

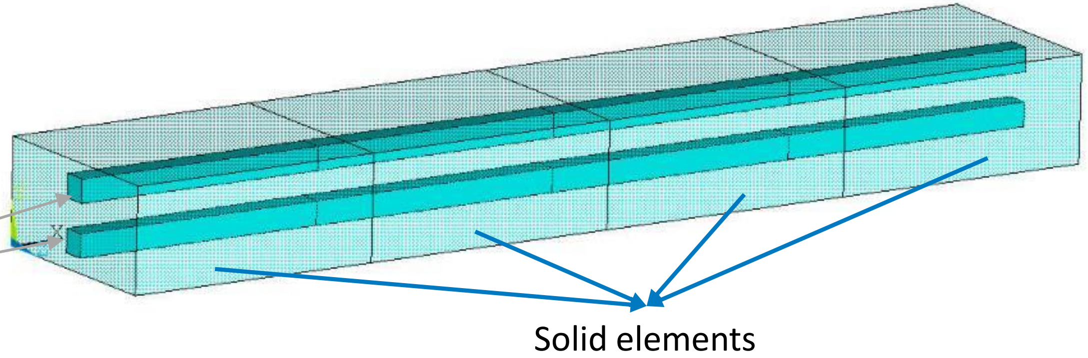
Assign a Global ID (GID) to selected MESH200 elements

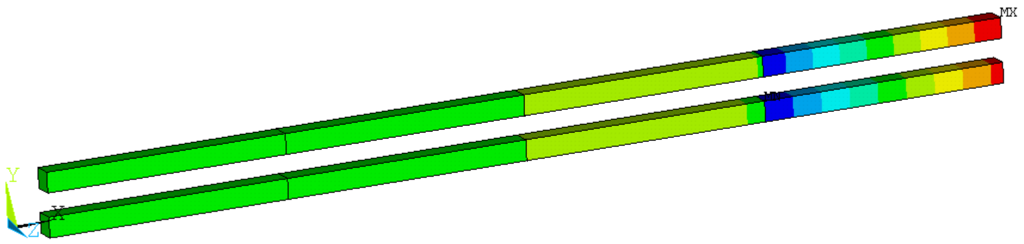
EREINF command assigns the Global ID to reinforcing members

- Select the reinforcing members by GID: EMSEL
- Use pre-/post-processing commands

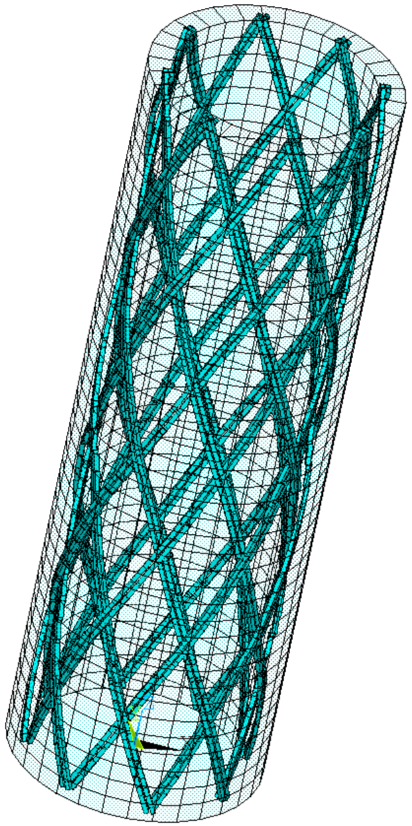
```
/prep7
…
! Assign Global ID to MESH200
esel,
EGID, 2
esel,
EGID, 3
esel,
EGID, 4
…
EREINF

! Select Members
esel, all
EMSEL,,,,2,4,2
EPLOT


/post1
! plot stress output
esel,,ename,,264
EMSEL,,,,2,4,2
plesol,s,x
```





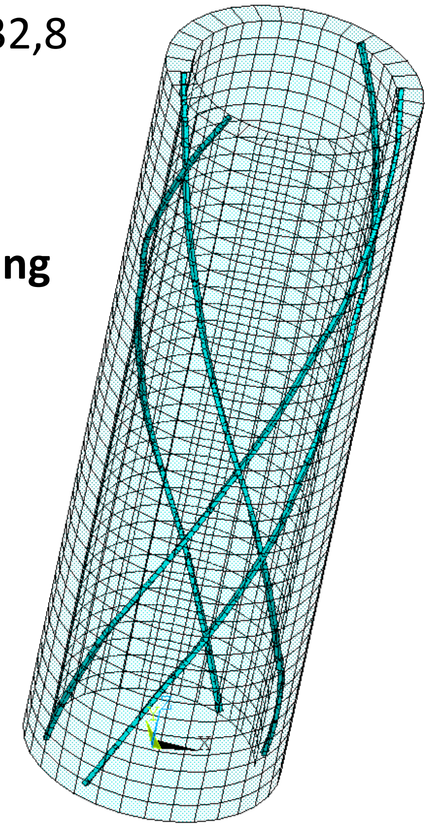


EMSEL vs LAYER command



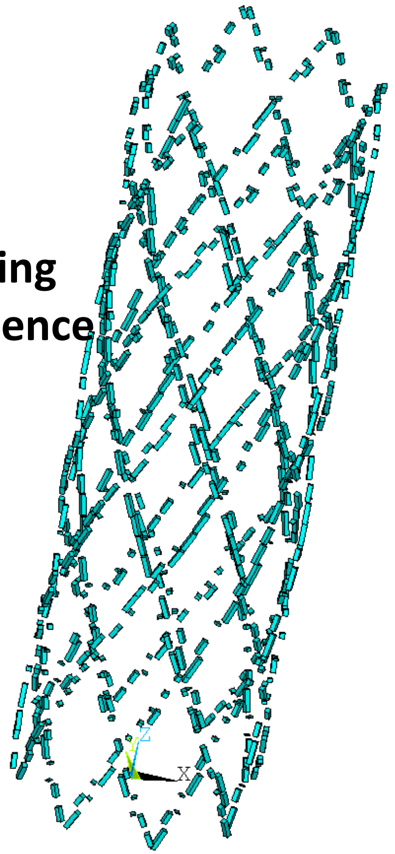
EMSEL,,,,1,32,8
/ESHAP,1
EPLOT

Selection reinforcing members by GIDs
→ More intuitive



LAYER,2
/ESHAP,1
EPLOT

Selection reinforcing members by sequence number (local)

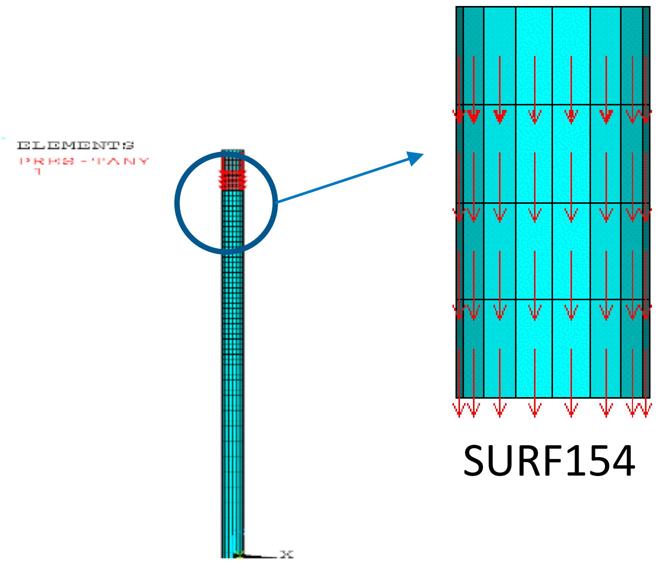


General Distributed Load on Structural Solid Elements

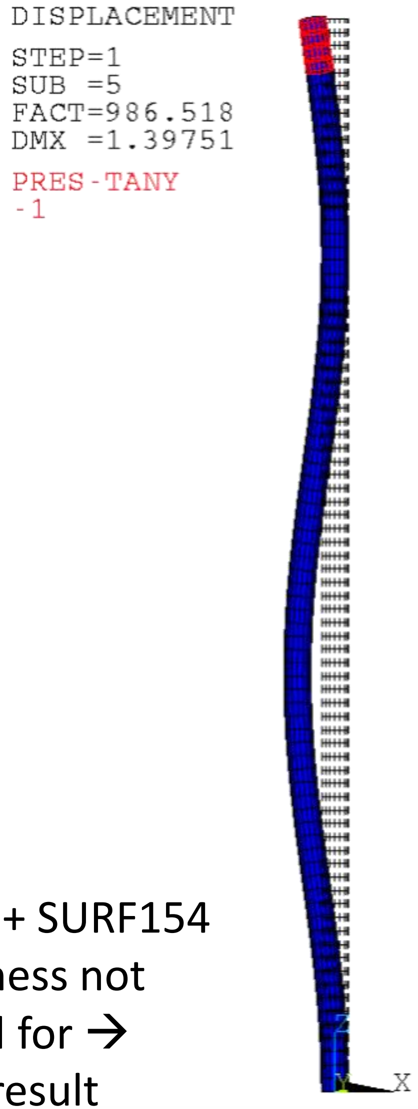
- Before REV 2019 R1 solid elements are capable of normal and constant pressure only
- For general distributed load (normal, tangential, fixed direction, tapered, etc), surface effects element must be used
- Lack of direct general surface load on solid elements
  - Added complications to modeling
  - Difficult to use
  - Affected solution robustness and accuracy

- In REV 2019 R1 (beta), the following elements are supported for general surface loading:
  - SOLID185, SOLID186, SOLID187, SOLSH190, SOLID285

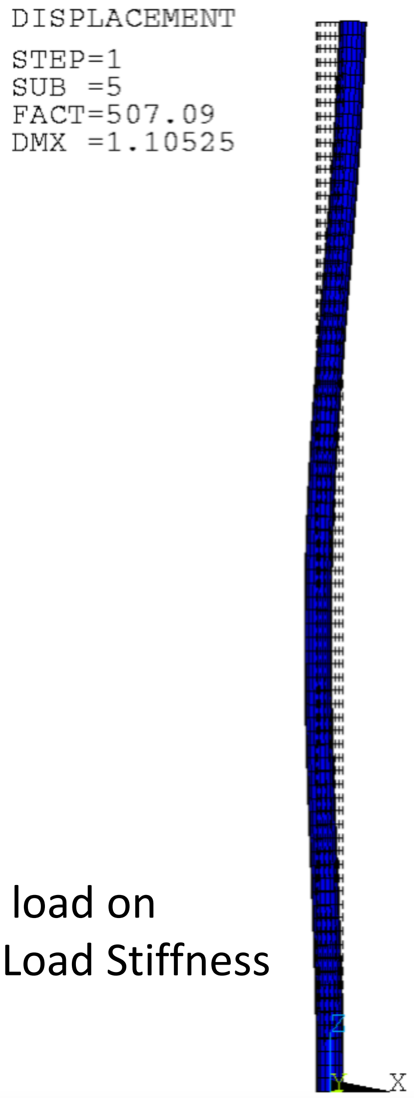
Eigenvalue Buckling of a Straight Pipe with Tangential Surface Load



SURF154

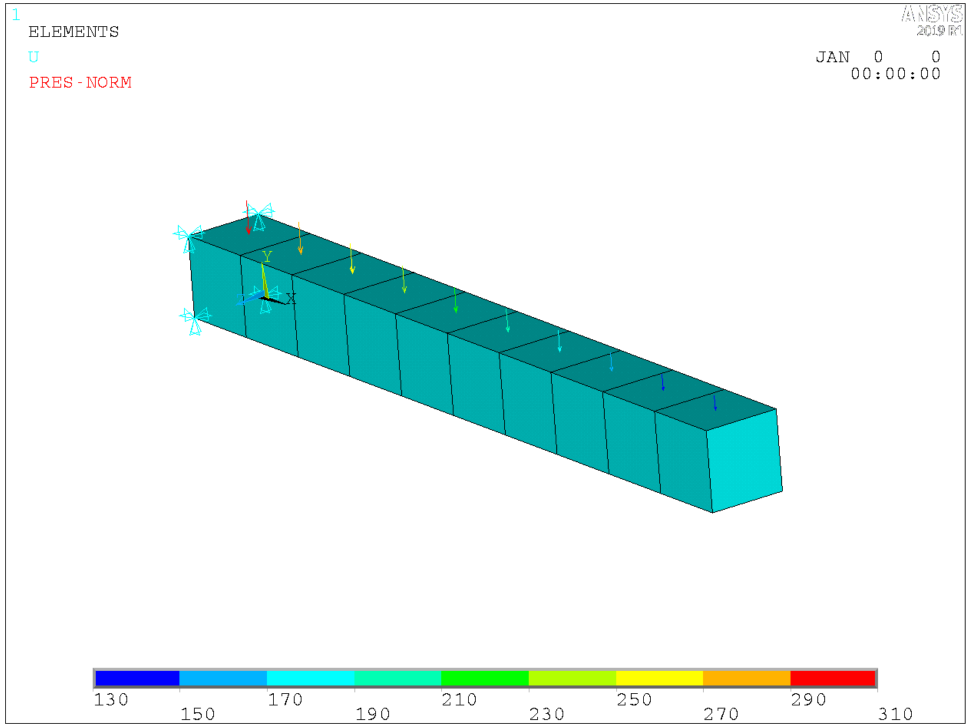


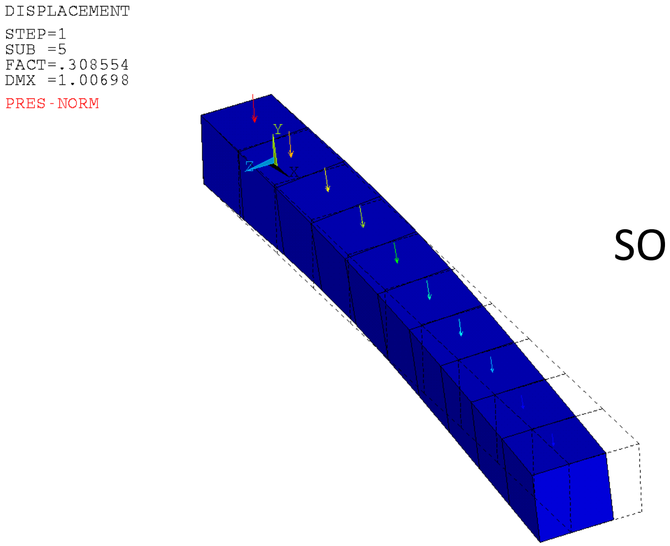
SOLID185 + SURF154
Load stiffness not
accounted for →
incorrect result



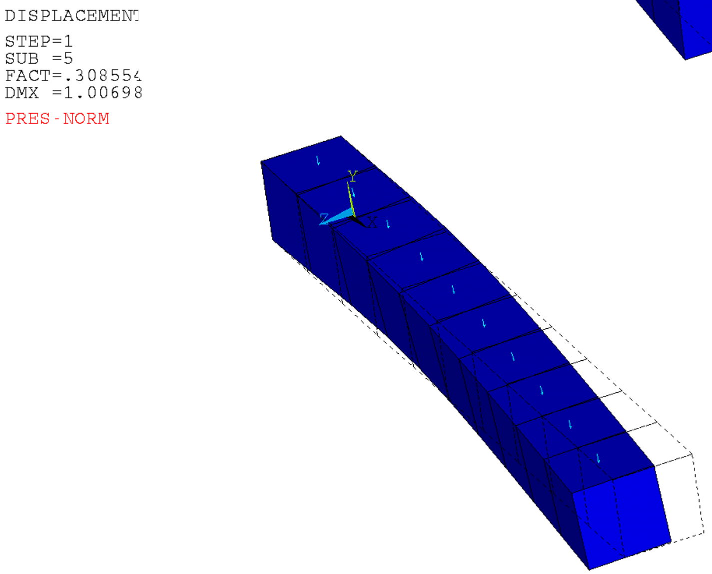
New tangential load on
SOLID185 with Load Stiffness
→ correct

Example 2: Global tapered pressure





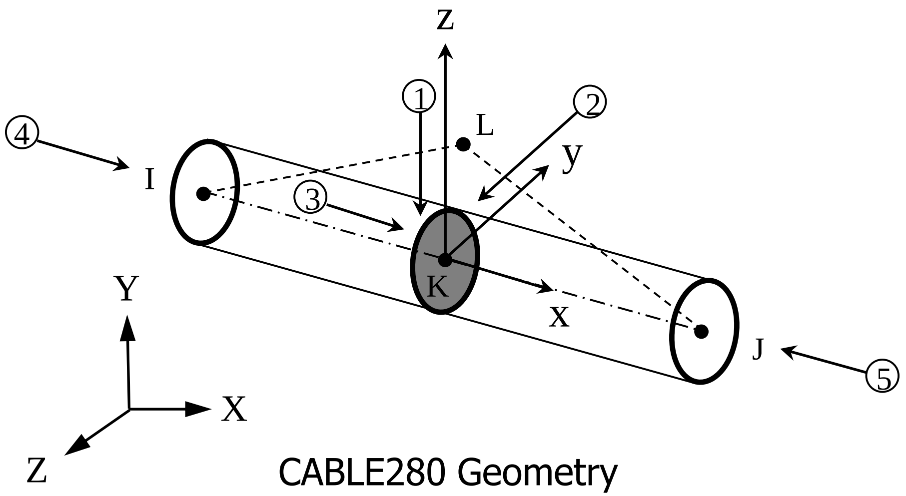
SOLID185 + SURF154



SOLID185 Only

CABLE280 Element (beta)

- Current ANSYS elements suitable for simulating cables: LINK180, BEAM188, BEAM189.
- Convergence difficulty for extremely flexible cables (e.g., undersea cables)
- Require fine mesh to achieve accurate solution in both displacements and axial force.



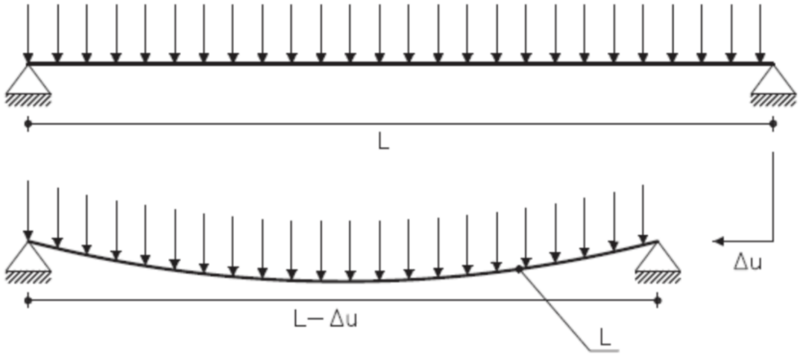
CABLE280 Geometry

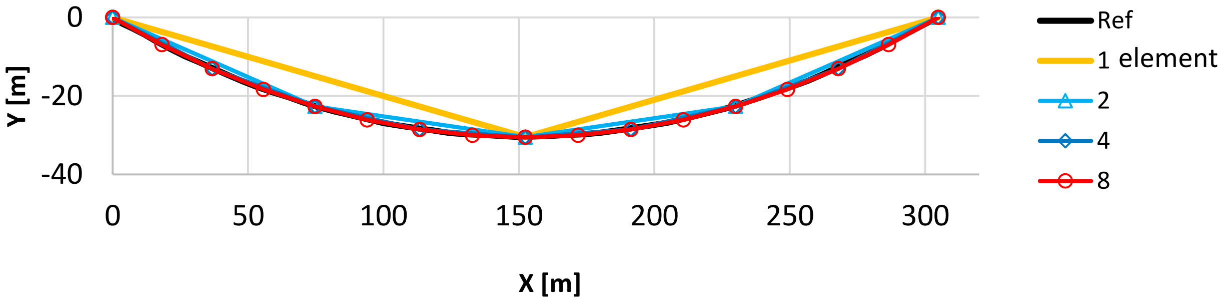
CABLE280

- Four nodes. Fourth orientation node required only for applying transverse load.
- Degree of freedom: UX, UY, UZ.
- Two integration points for stiffness, three integration points for mass and distributed loads.
- Mixed U/F formulation: quadratic approximation for displacements and linear approximation for axial forces.
- Axial force DOFs are incompatible and internal to the element.

CABLE280 Example #1: Suspension Cable

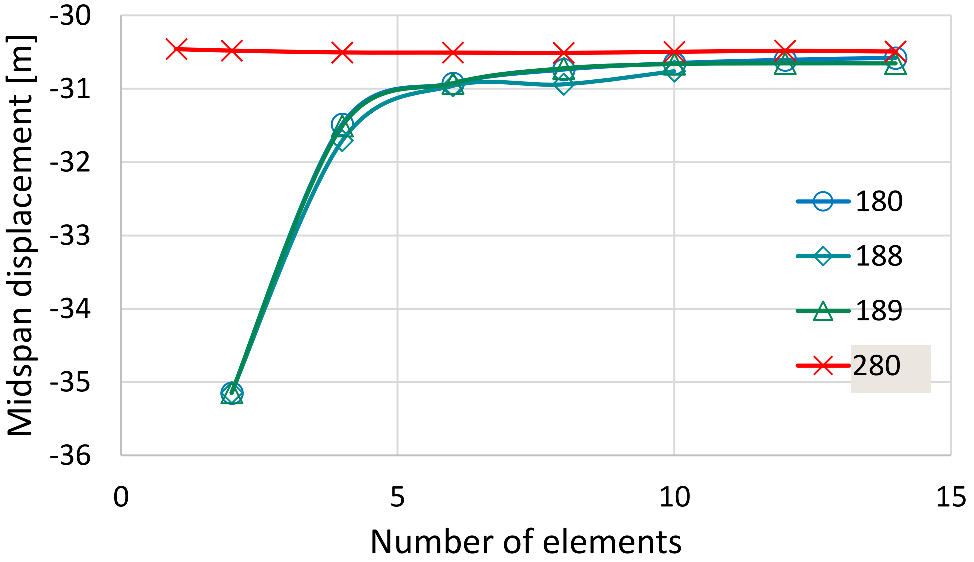
Cross-section area 548.4 mm^2, elastic modulus 131 GPa, length 312.73 m.





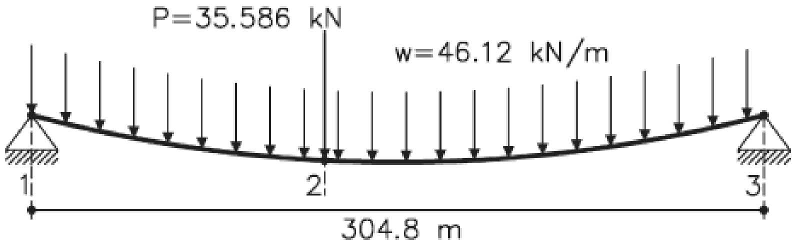
Start from a straight horizontal reference configuration, apply a prescribed displacement on the right support $\Delta u_x = -7.93$ m and gravity.

High accuracy even with a coarse mesh: relative error is 0.7% with one element.



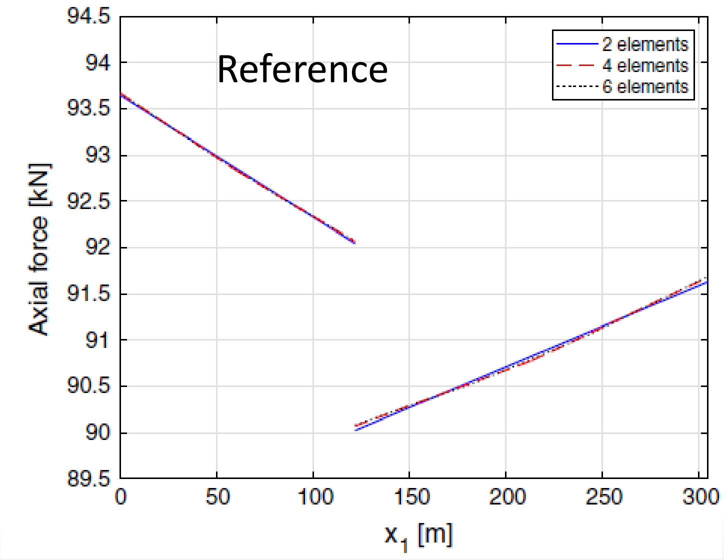
CABLE280 Example #2: Suspension Cable with a point load

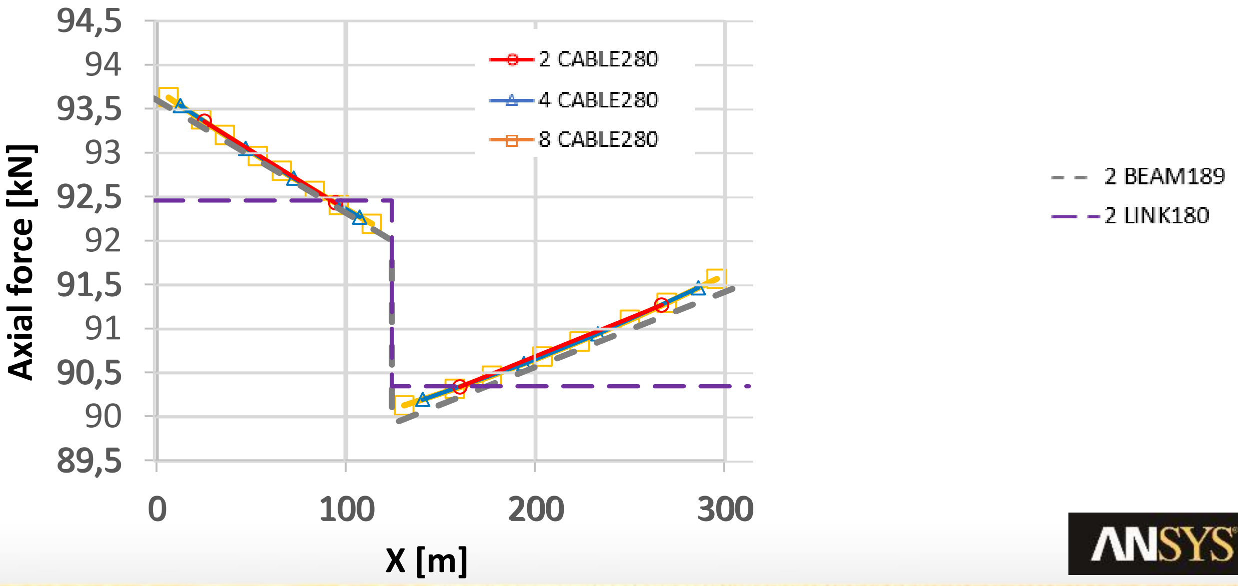
Cross-section area 548.4 mm^2, elastic modulus 131 GPa, length 312.73 m.



Start from a straight horizontal reference configuration, apply a prescribed displacement on the right support $\Delta u_x = -7.93$ m, gravity, and a point load at 2/5 of the cable span.

CABL280 is able to capture the jump in the axial force where a concentrated force applied, with coarse mesh.

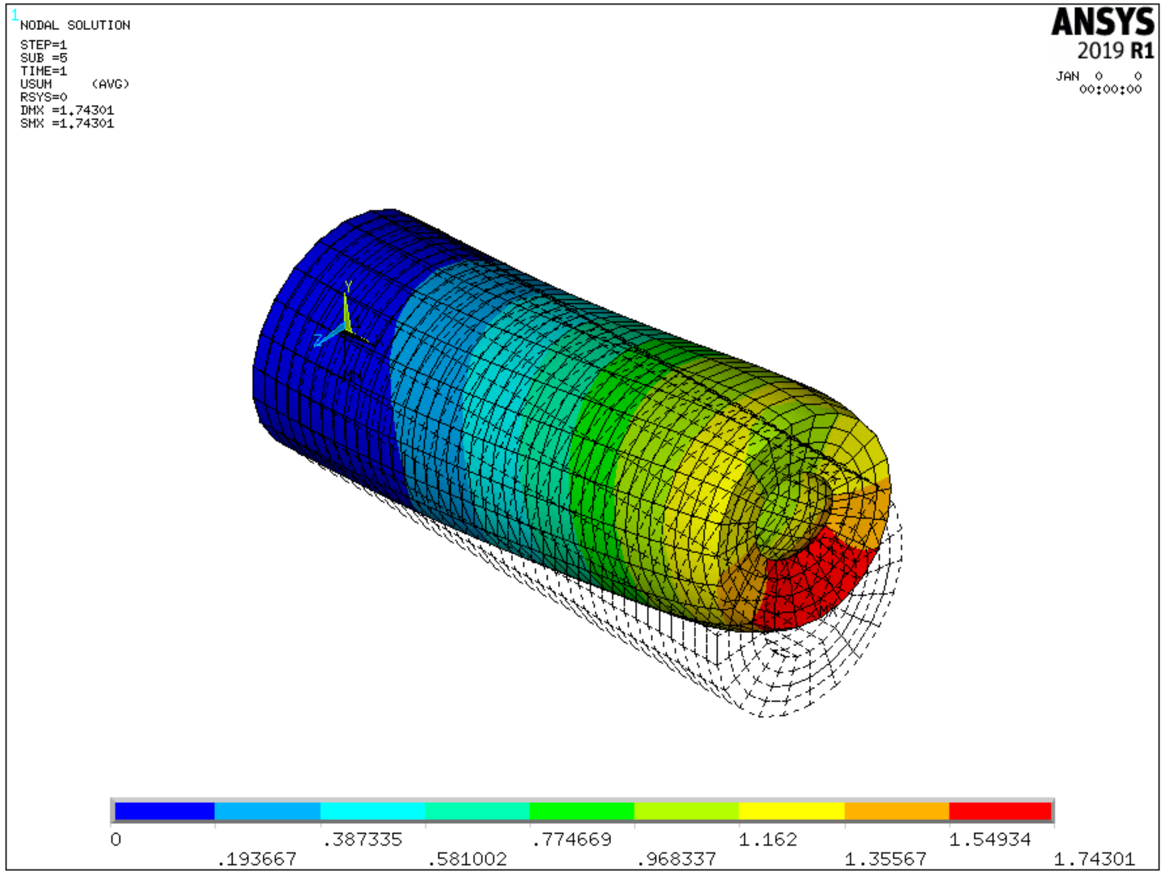




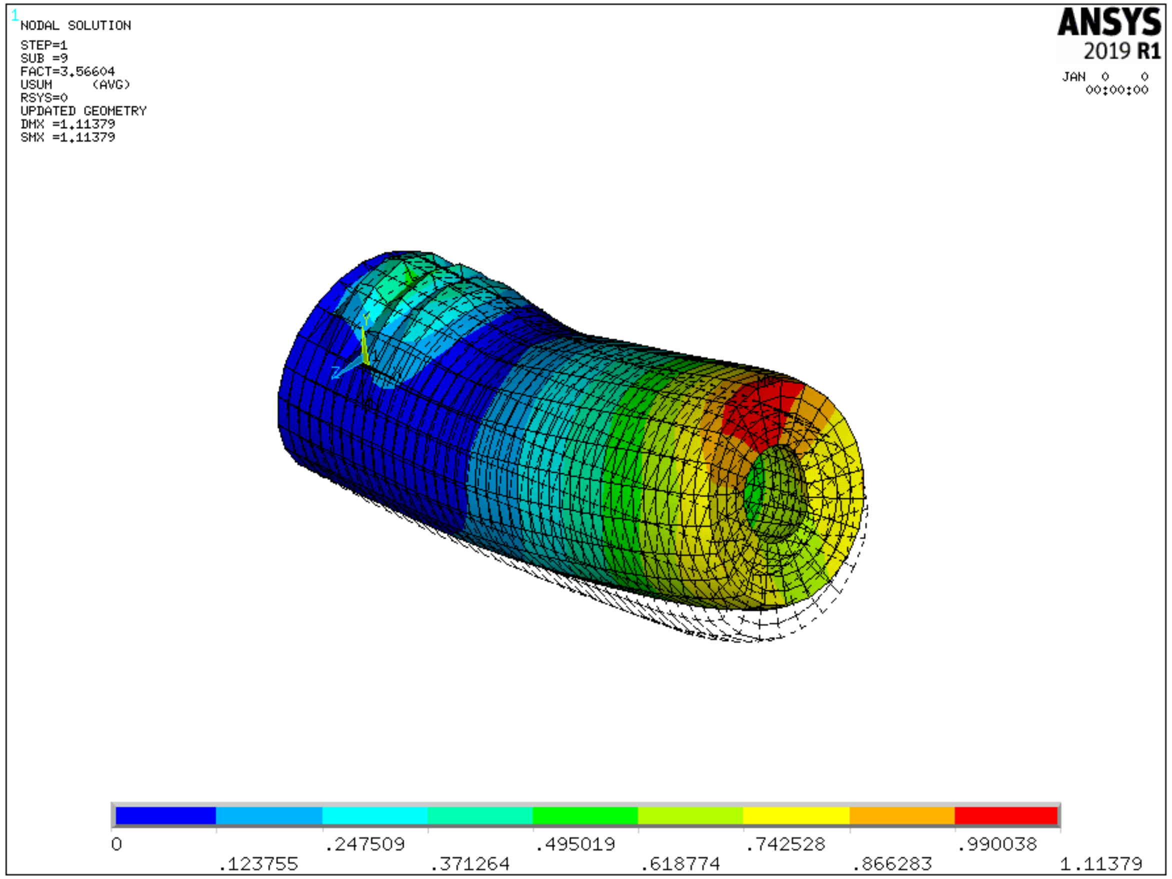
Linear Perturbation for Axisymmetric Structures under General 3D loads

- General axisymmetric solid elements SOLID272 / 273 are powerful to simulate axisymmetric structures under asymmetric loads.
- Due to the lack of Linear Perturbation (LP) support to the elements, the application of elements in linear dynamic analysis based on large deformation or deflections was limited.
- SURF159 is supported for applying various surface loads on SOLID272/273
- All LP analysis types supported: modal, Eigenvalue buckling, full Harmonic ..
- Supports both linear and tangent material stiffness options

SOLID273 Liner Perturbation Buckling Analysis

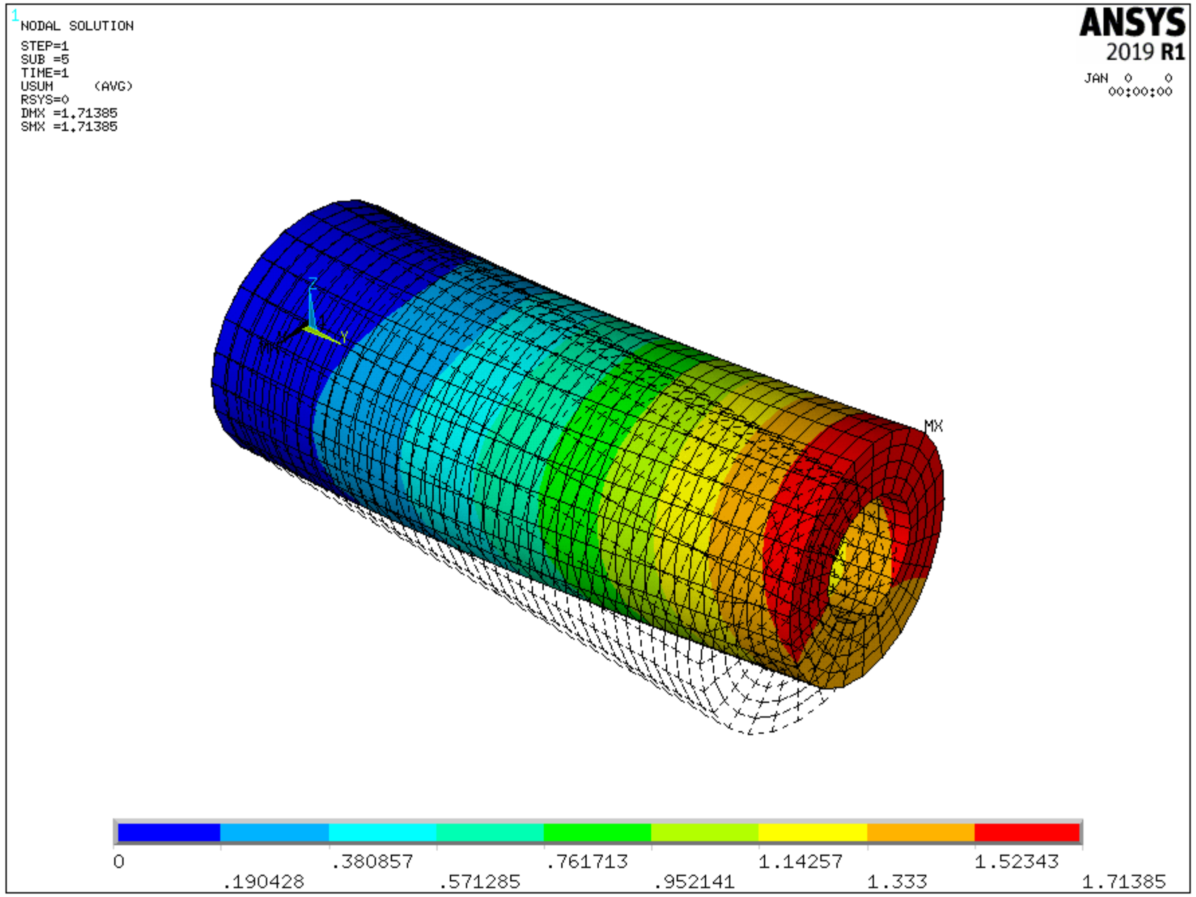


Deformed Shape after nonlinear static analysis

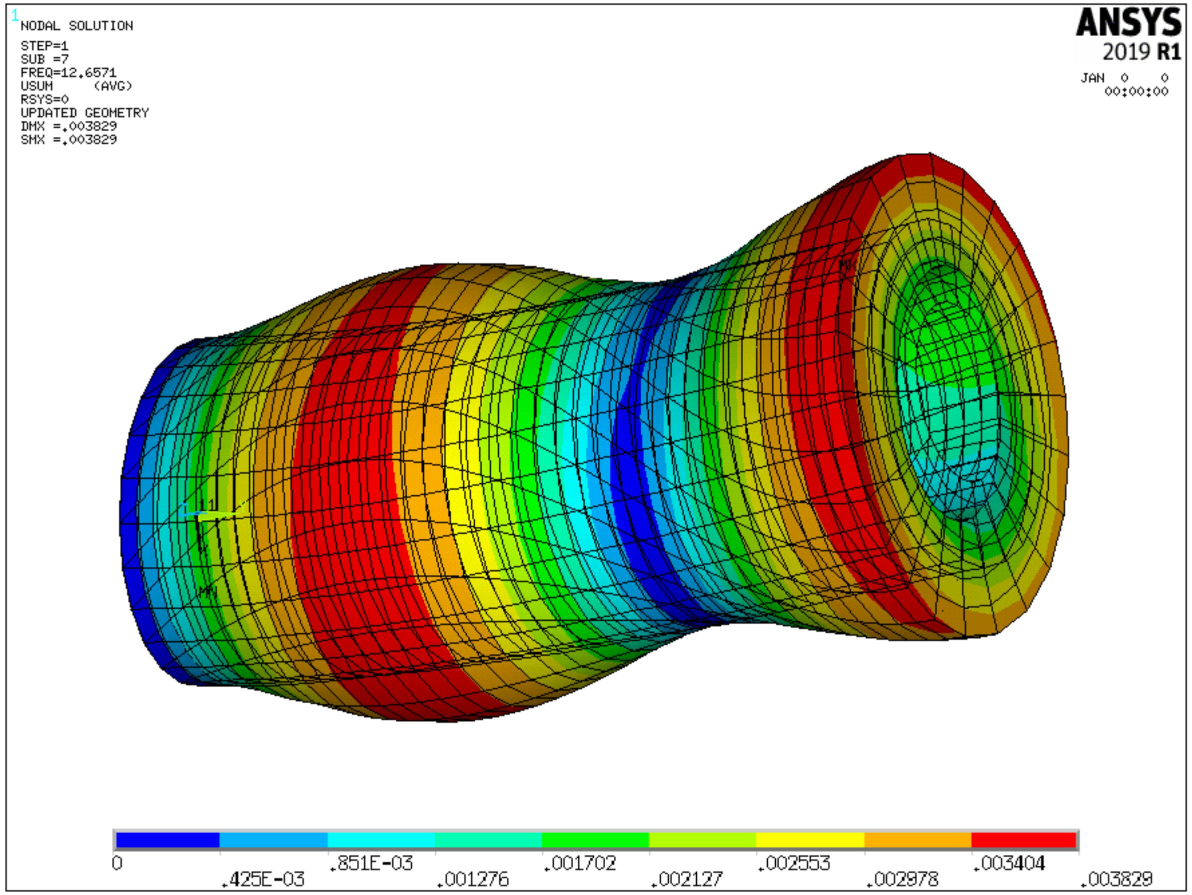


Subsequent LP Eigenvalue buckling mode

SOLID273/SURF159 Linear Perturbation Modal Analysis



Deformed Shape after nonlinear static analysis



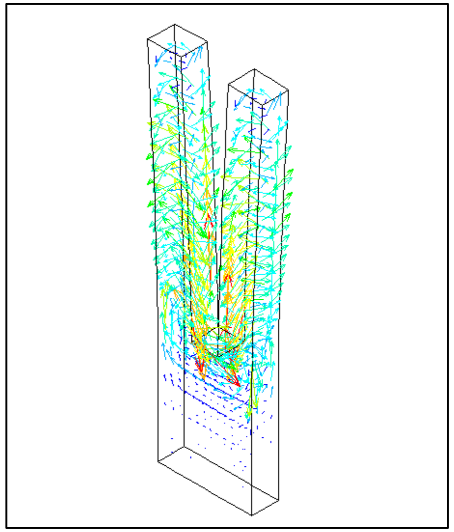
Subsequent LP Pre-stressed modal analysis based on the nonlinear results

Piezoelectric Analysis Enhancements

Dynamic piezoelectric analyses that use elements PLANE223, SOLID226, and SOLID227:

- Anisotropic elastic and dielectric losses
  - Input using new material tables:
    - TB,AVIS – Anisotropic viscosity
    - TB,ELST – Anisotropic elastic loss tangent
    - TB,DLST – Anisotropic dielectric loss tangent
  - For the simulation of loss anisotropy in bulk and surface acoustic wave devices
- Heat generation rate (JHEAT) due to the combination of structural and electric losses
  - For a subsequent thermal analysis to predict the heating of a piezoelectric devices due to structural and electric losses
- New output item: Poynting vector
  - For power flow visualization in piezoelectric devices

Poynting Vector in the Tuning Fork



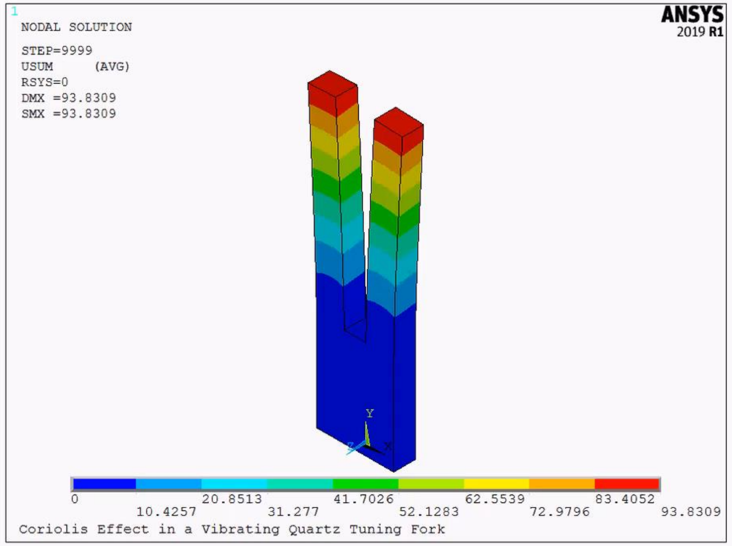
Piezoelectric Vibrations with Anisotropic Structural Loss

Piezoelectric harmonic analysis of quartz crystals with structural losses

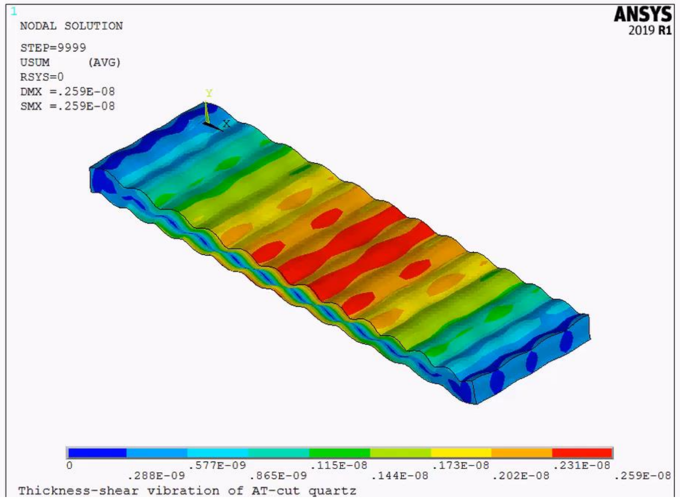
- **Losses are modeled using the anisotropic viscosity table (TB,AVIS)**
- **Different modes of vibration have different loss factors**

```
! Viscosity constants for Quartz, N/m**2 s
eta11= 1.37e-3
eta12= 0.73e-3
eta13= 0.72e-3
eta14= 0.01e-3
eta33= 0.97e-3
eta44= 0.36e-3
eta66= 0.32e-3

! Anisotropic viscosity table
tb,AVIS,1,,,0
tbda,1,eta11,eta12,eta13,,eta14
tbda,7,eta11,eta13,,-eta14
tbda,12,eta33
tbda,16,eta66,,eta14
tbda,19,eta44
tbda,21,eta44
```

Tuning fork quartz crystal with Coriolis effect vibrating in FLEXURE mode at 32.768 kHz



AT-cut quartz plate vibrating in THICKNESS SHEAR mode at 1664 kHz

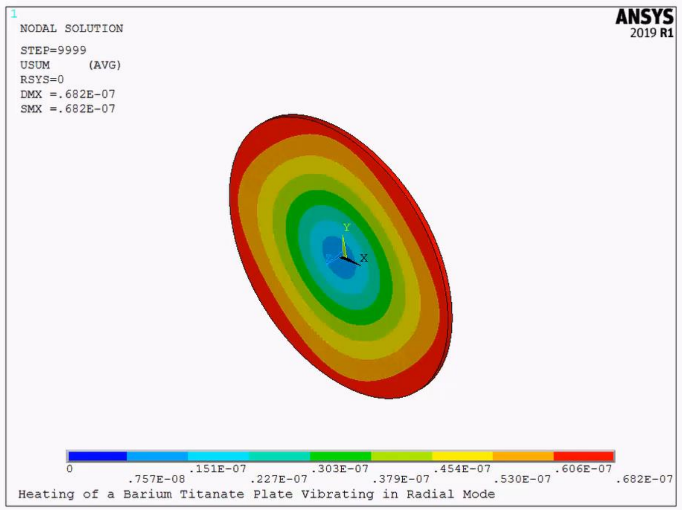


Heating of a Piezoelectric Disc due to Anisotropic Electric Losses

Piezoelectric harmonic analysis of a Barium Titanate disc vibrating in radial mode at 263 kHz

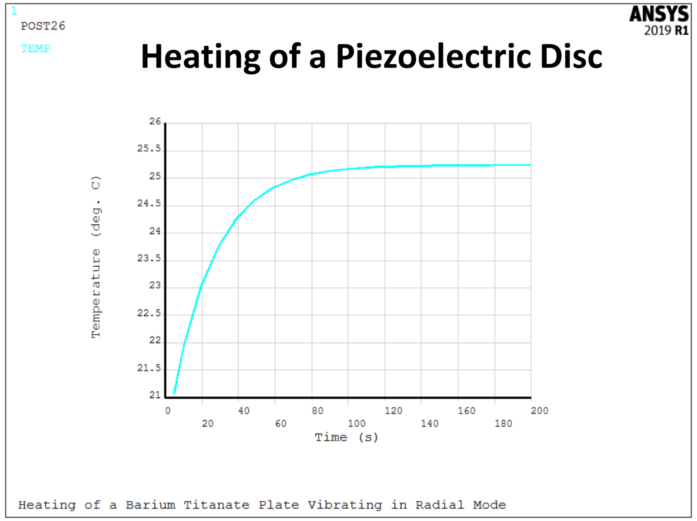
- **Electric losses are modeled using the dielectric loss tangent table (TB,DLST)**
- **Time-averaged heat generation due to these losses is stored as JHEAT and transferred to a thermal analysis via LDREAD,HGEN**

Barium Titanate Disc vibrating in RADIAL mode at 263 kHz



- **Thermal transient analysis of the disc is performed to determine the temperature rise due to dielectric heating**

```
! Loss tangent coefficients in radial and axial
directions
tand11=0.005
tand33=0.009

! Dielectric loss tangent table
TB,DLST,1
TBDATA,1,tand11,tand11,tand33
```



Material Designer

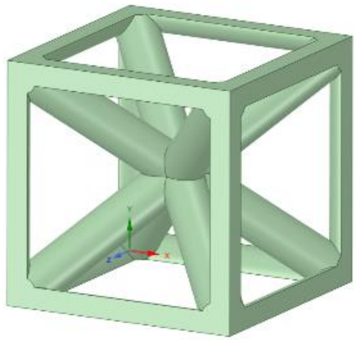
ANSYS 2019 R1 update

Content

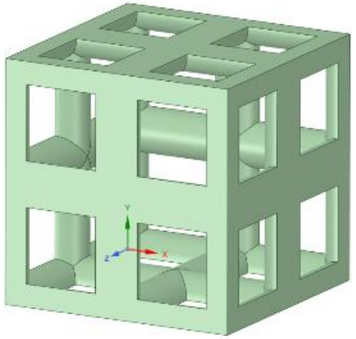
- Additional Lattice Structures
- Non-Uniformly Distributed Chopped Fiber Composites
- UI Improvements

Additional Lattice Structures

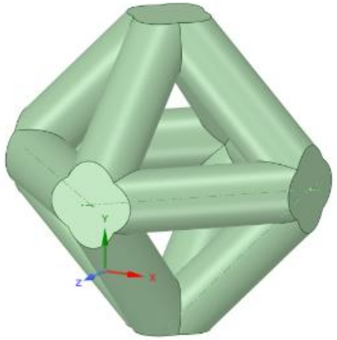
- Additional Predefined Lattice Structures



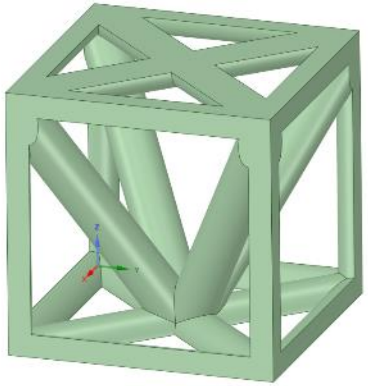
Cubic with center supports



Cubic with side cross supports

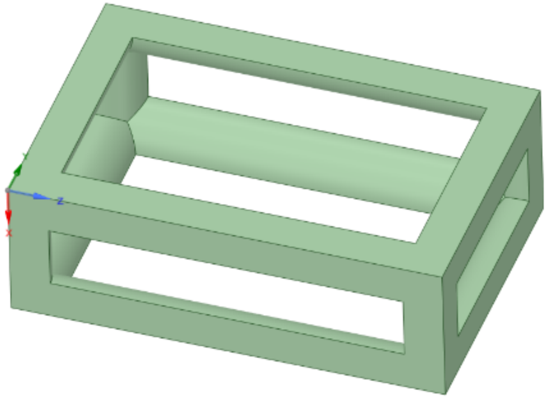


Double pyramid



Cubic with bottom center supports

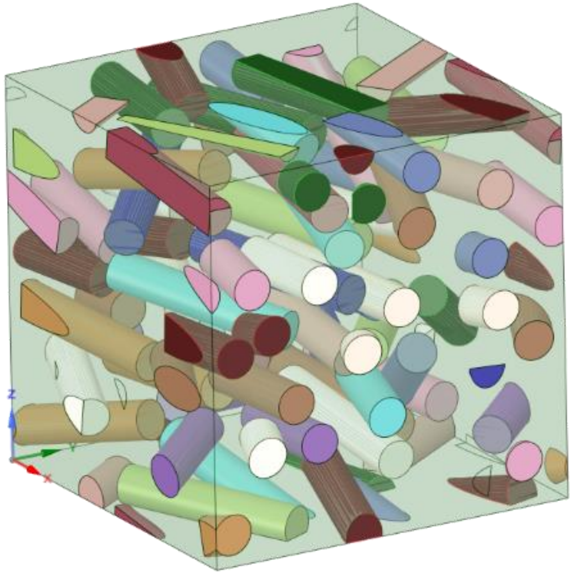
- User Defined Lattice Structures
  with a Rectangular Cuboid as Unit Cell
  i.e. different sizes in each direction



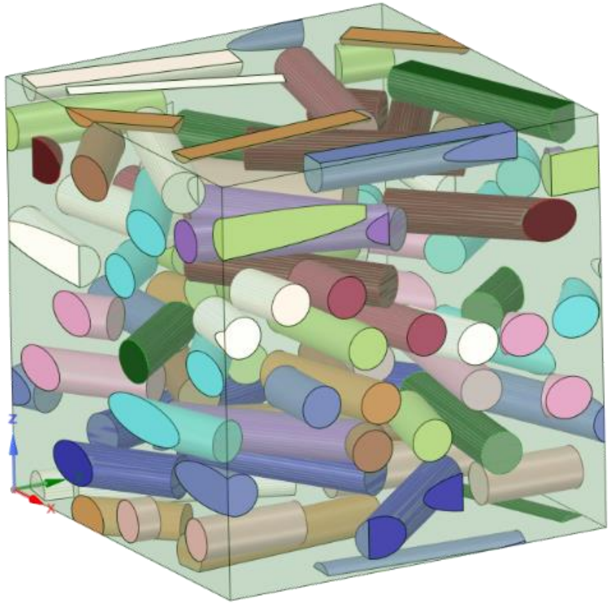
Non-Uniformly Distributed Chopped Fiber Composites

Specify a target orientation tensor

In particular, this allows to generate RVEs, in which fibers are



mostly aligned with the X axis



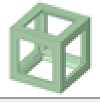
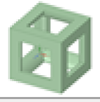
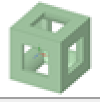
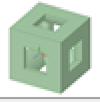
or oriented parallel to the XY plane

UI Improvements

- Easily Access the (Solver) Log Files

Logs	
RVE log	
Solver logs	

- See an Image of each Variation (for variable materials)

Name	Values[0]	Values[1]	Values[2]	Values[3]
Image				
Parameters				
Volume Fraction	0.2	0.3	0.4	0.5

- Easily Access the Material Designer User's Guide

Open

Help

Mechanical Topology Optimization

ANSYS 2019R1 update

Outline

Support thermal compliance objective and Temperature constraint optimization of Steady State Thermal system

Reload Volume Fraction from last optimized iteration

Support stress constraints in regions outside the optimization region

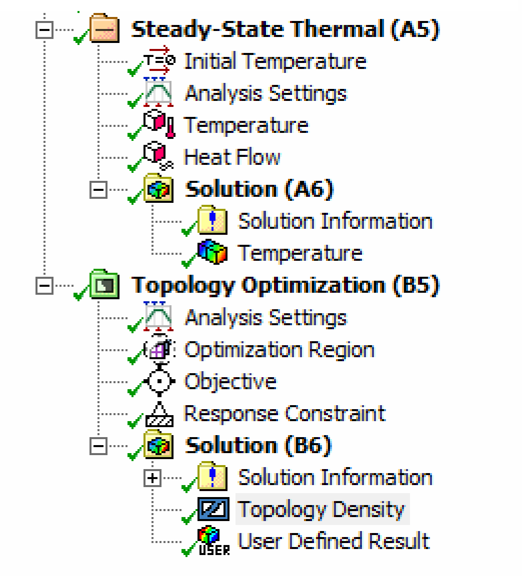
Support optimization of shell bodies

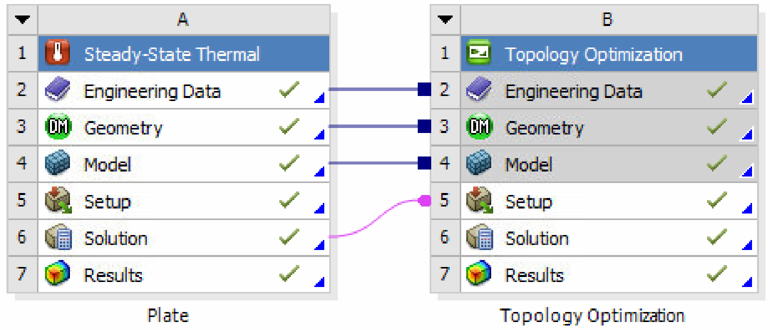
Support Smoothing Result for Smoothed STL

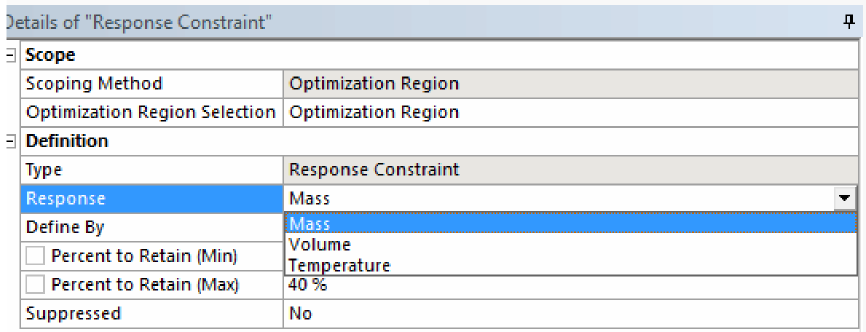
Optimization of Steady State Thermal System

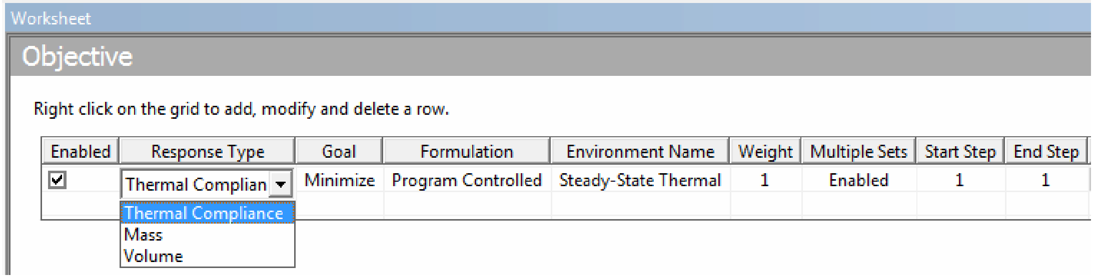
Thermal Compliance objective and Temperature Constraint associated to Steady State Thermal System is supported

Mass and Volume is supported as Optimization objective/constraint



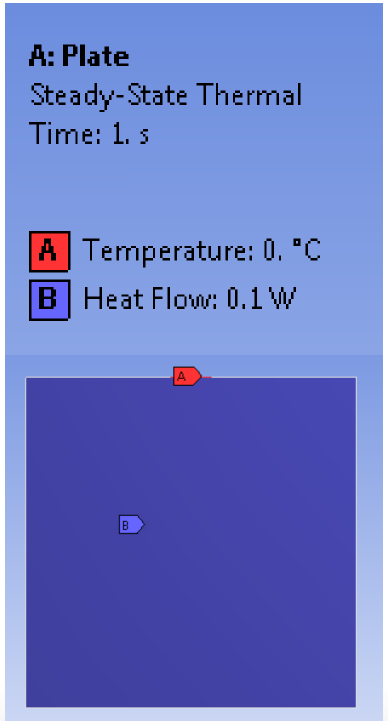


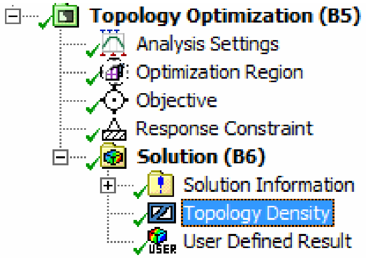




Optimization of Steady State Thermal System

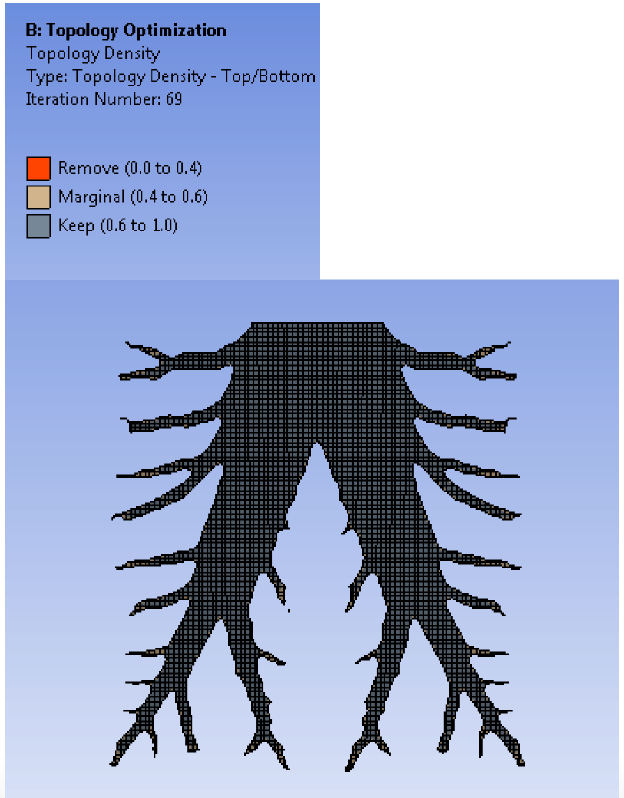
For the plate model, Temperature and Heat Flow is applied to Steady State Thermal System; Thermal compliance is chosen as Objective and Response constraint of type Mass is chose with Range specified between 1 and 40 percent. The topology density result shows the optimized topology with mass percentage as 34.797 percent, which will achieve the maximum heat transfer





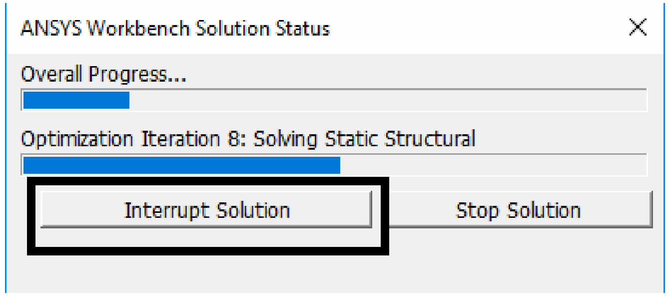
Details of "Topology Density"

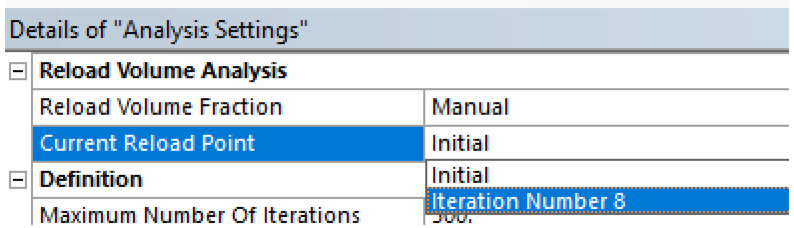
Scope	
Scoping Method	Optimization Region
Optimization Region	Optimization Region
Definition	
Type	Topology Density
By	Iteration
Iteration	Last
Retained Threshold	0.5
Exclusions Participation	Yes
Calculate Time History	Yes
Suppressed	No
Results	
Minimum	2.0813e-002
Maximum	1.
Average	0.39544
Original Volume	2.5e+009 mm³
Final Volume	8.6993e+008 mm³
Percent Volume of Original	34.797
Original Mass	19.625 t
Final Mass	6.8289 t
Percent Mass of Original	34.797

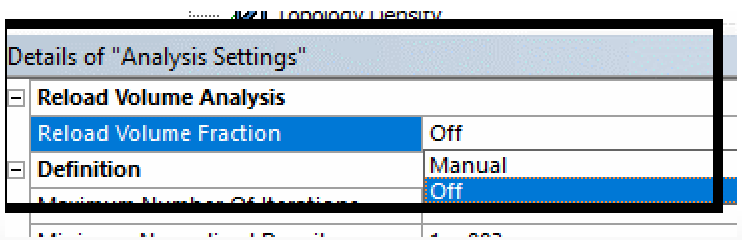


Reload Volume Fraction from last optimized iteration

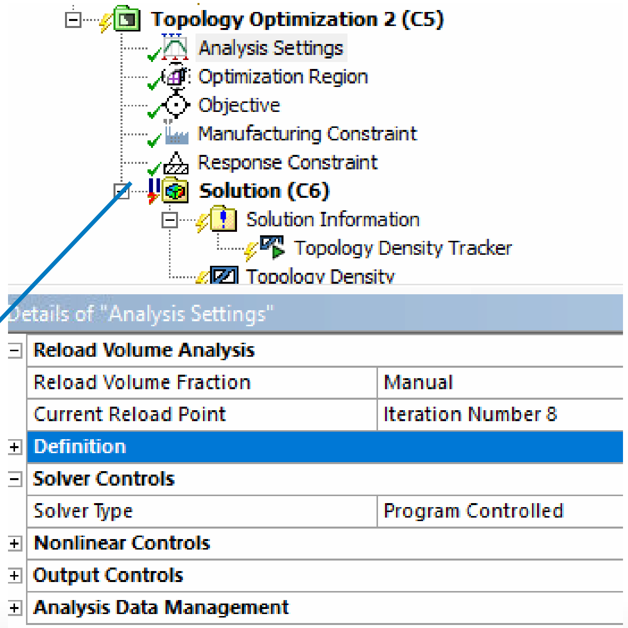
Reload Volume Fraction from last iteration to continue optimization with modification of Objectives and Response Constraints. Solve or Interrupt the optimization run, set the Reload Volume Fraction to Manual, Pick the Current Reload point based on last solved iteration, modify objective or constraints and perform optimization. This will continue optimization by reloading the volume fraction computed in the previous run





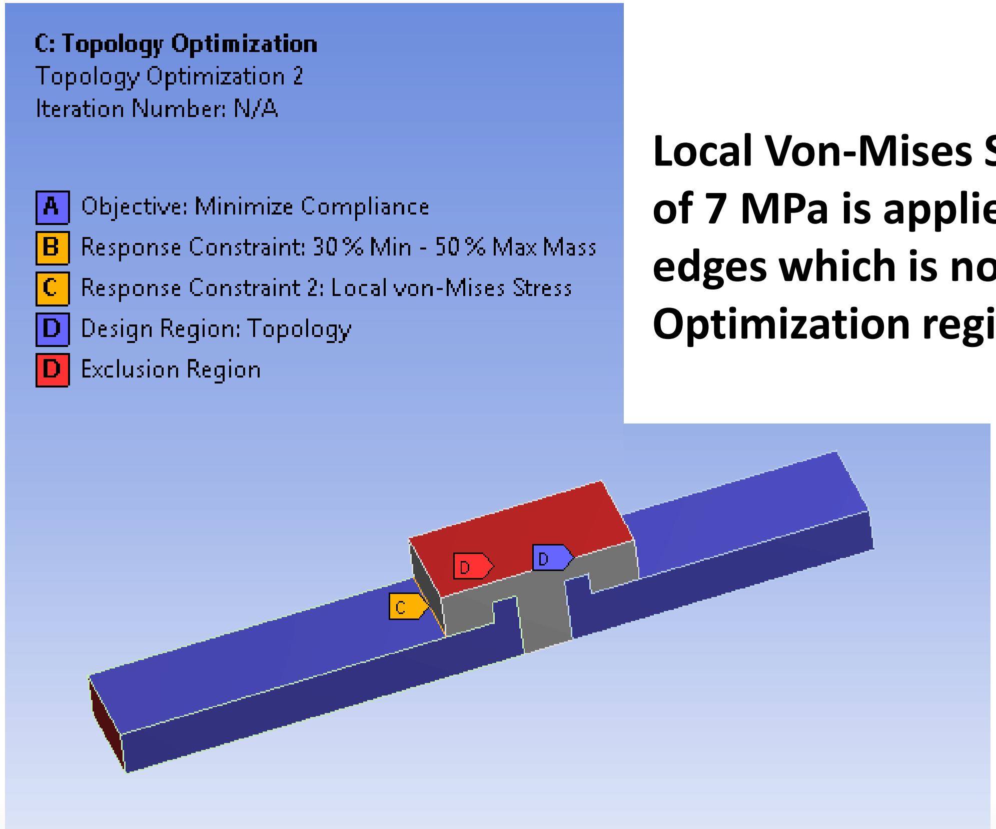


Icon to indicate the Reload Volume Fraction option

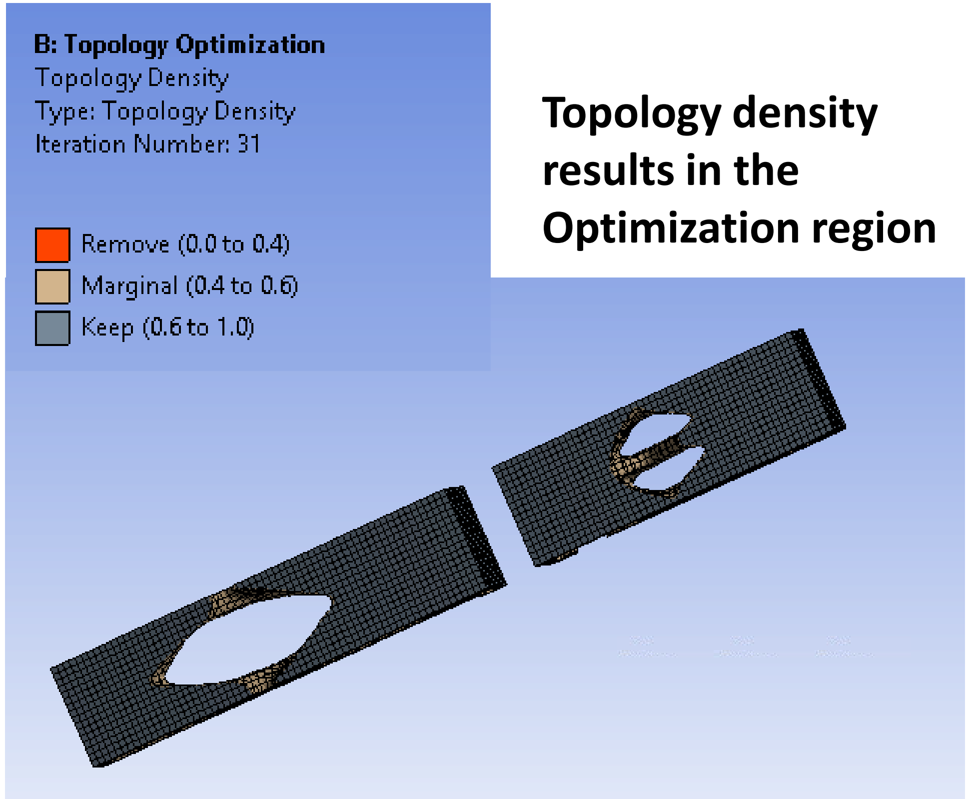


Stress constraint and Shell body optimization

Support stress constraints in the region outside of Optimization region. It could be exclusion region or other parts of the entire model.



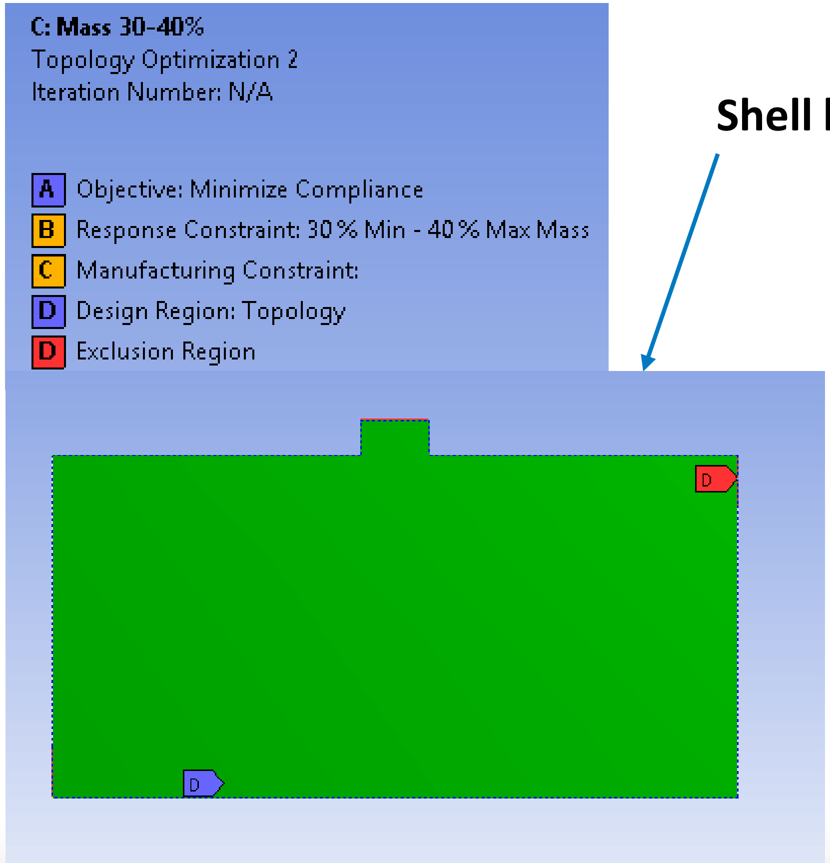
Local Von-Mises Stress constraint of 7 MPa is applied to connector edges which is not part of Optimization region



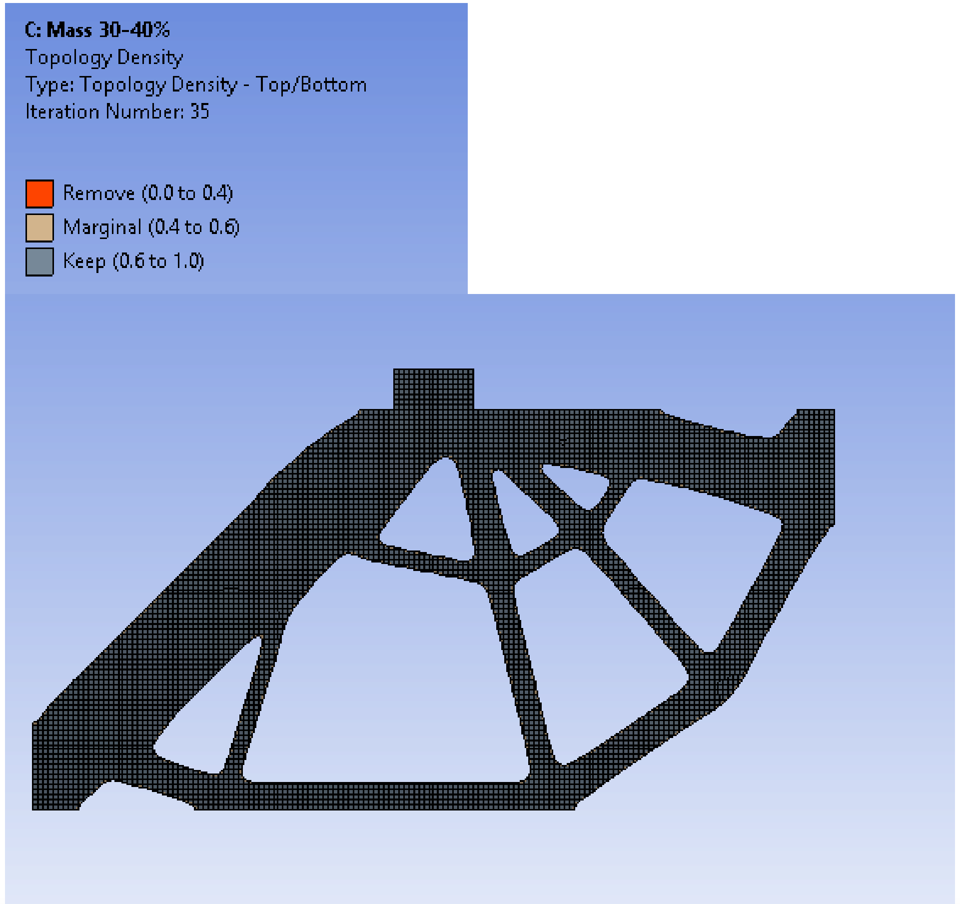
Topology density results in the Optimization region

Optimization of Shell bodies

Shell bodies are also optimized if included in the optimization region, but it is not optimized in the thickness direction

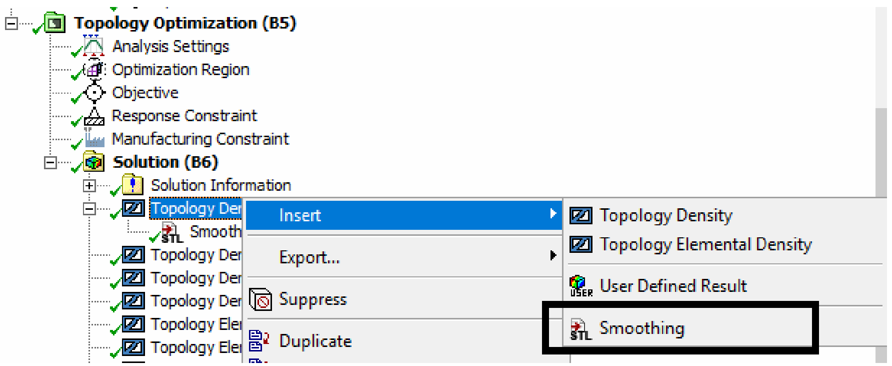


Shell body



Smoothing Result

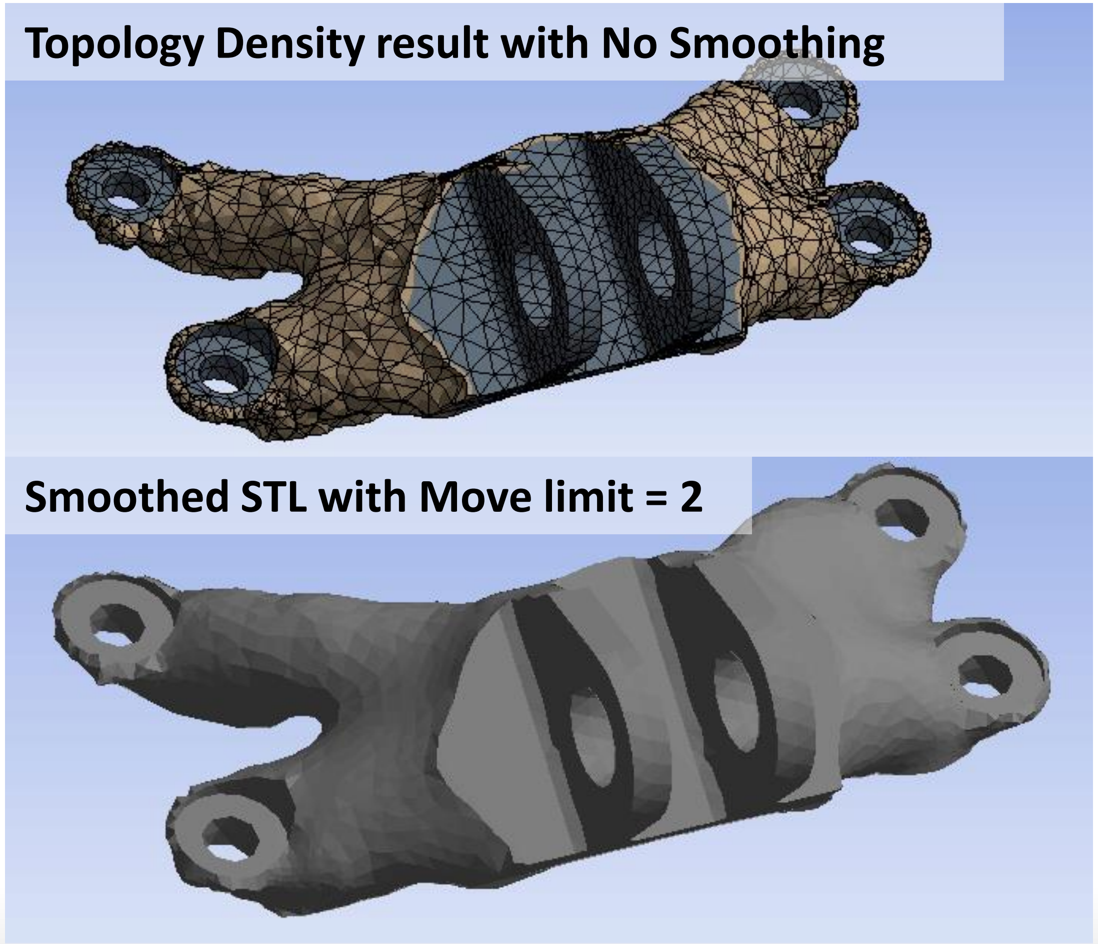
Smoothing result can be added under Topology Density result. It creates Smoothed STL, which can then be used for design validation study



Move limit default is 0. The STL is smoothed further with increasing value of Move limit

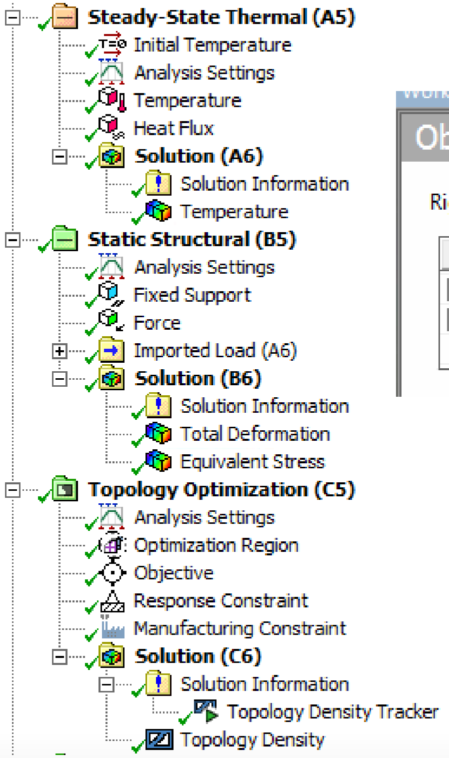
Details of "Smoothing"

Definition	
Move Limit	2.
File Name	C:\Users\jchopra\AppData\Local\Temp
Export Model (Beta)	No
Display	
Color	
Statistics (Triangles)	8498



Thermal Stress Optimization (Beta)

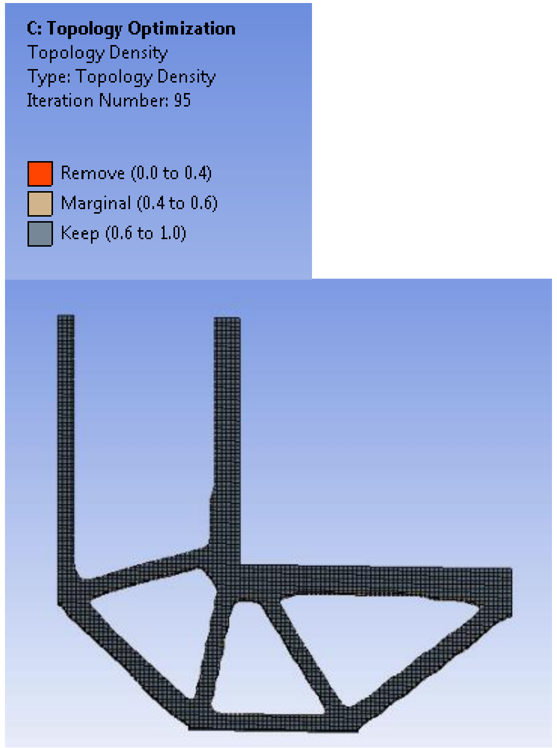
Support thermal compliance objective for Steady State Thermal System and Compliance objective for Static Structural system to optimization thermal stress case. Temperature constraint on Steady State Thermal System and Response constraint like Stress on Static Structural system can be added at the same time to Topology optimization



Objective

Right click on the grid to add, modify and delete a row.

Enabled	Response Type	Goal	Formulation	Environment Name	Weight	Multiple Sets	Start Step	End Step
☑	Compliance	Minimize	Program Controlled	Static Structural	1	Enabled	1	1
☑	Thermal Compliance	Minimize	Program Controlled	Steady-State Thermal	1	Enabled	1	3



In this problem, both thermal compliance for Steady state thermal system and compliance for static structural is applied with Volume response constraint of 30 percent

Mechanical Level-Set based Topology Optimization –BETA

ANSYS 2019R1 update

Outline

Mesh

- ➢ Support Tetra mesh (linear or quadratic) for the optimizable regions. No restriction for non optimizable regions.

Static Linear Analysis

- ✓ Support the following BC: fixed parts, prescribed displacement
- ✓ Support the following Loads: surface load, nodal load, volume load (acceleration, gravity, etc)
- ✓ Available criterion: “generalized” compliance (both valid for standard loads and prescribed displacement)

Modal Analysis

- ✓ support the following criterion: eigen frequency

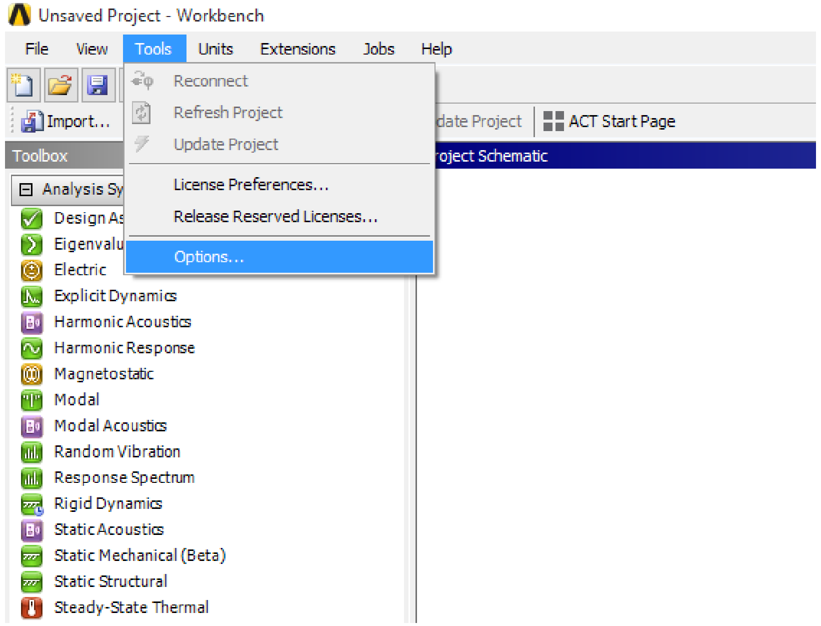
Geometric analysis

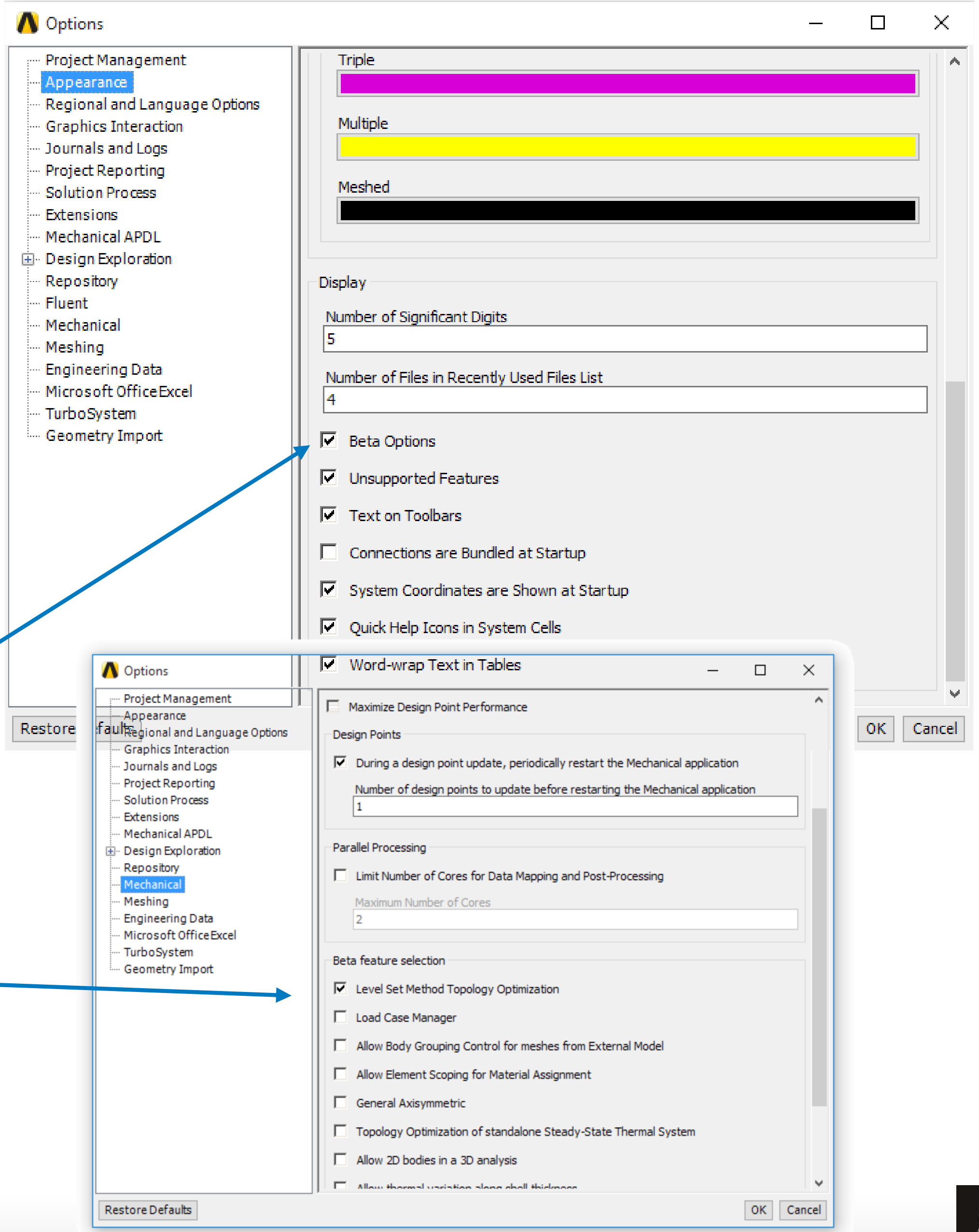
- ✓ Support volume, mass

Optimization

- ✓ Each criterion can be handled as an objective or a constraint

Activate the option

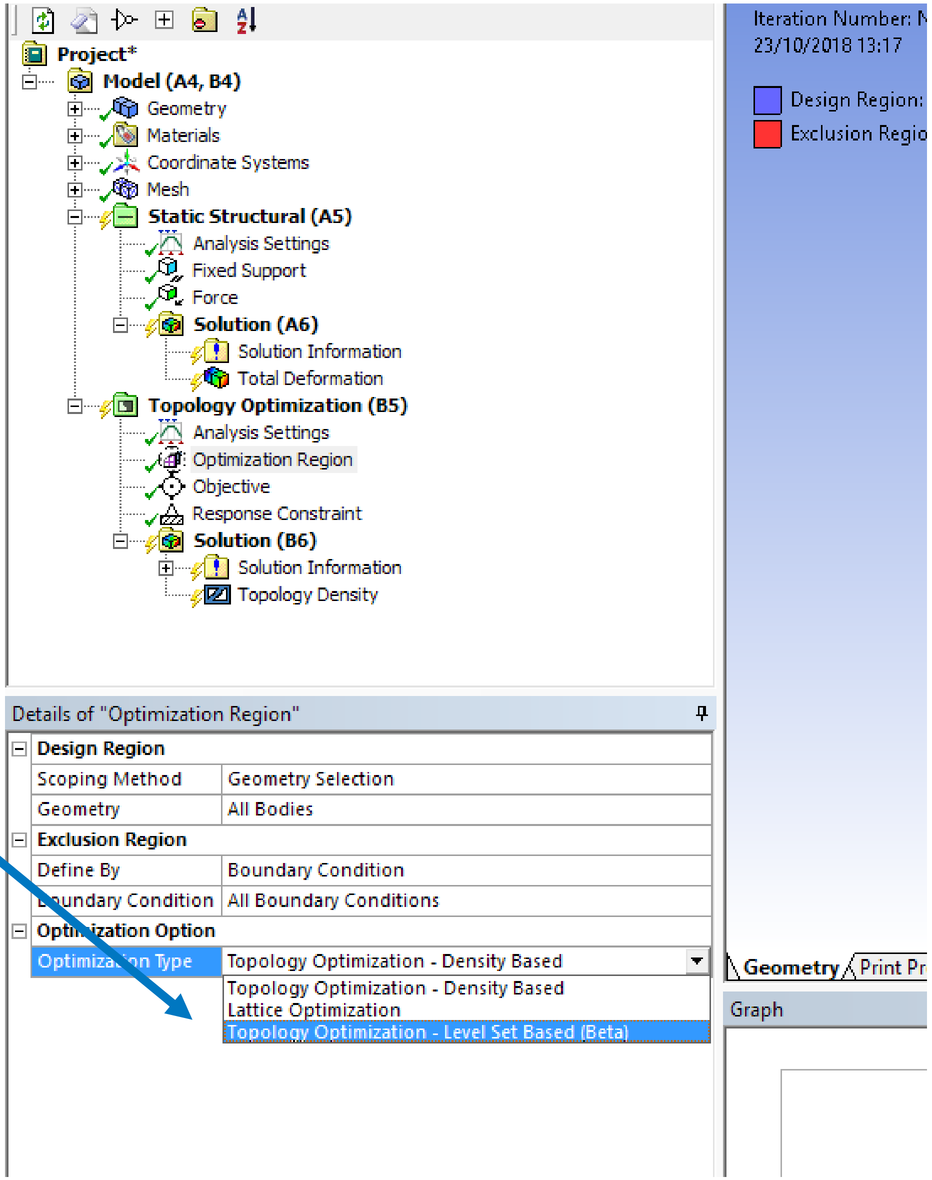




- Activate beta version
- And the level-set method

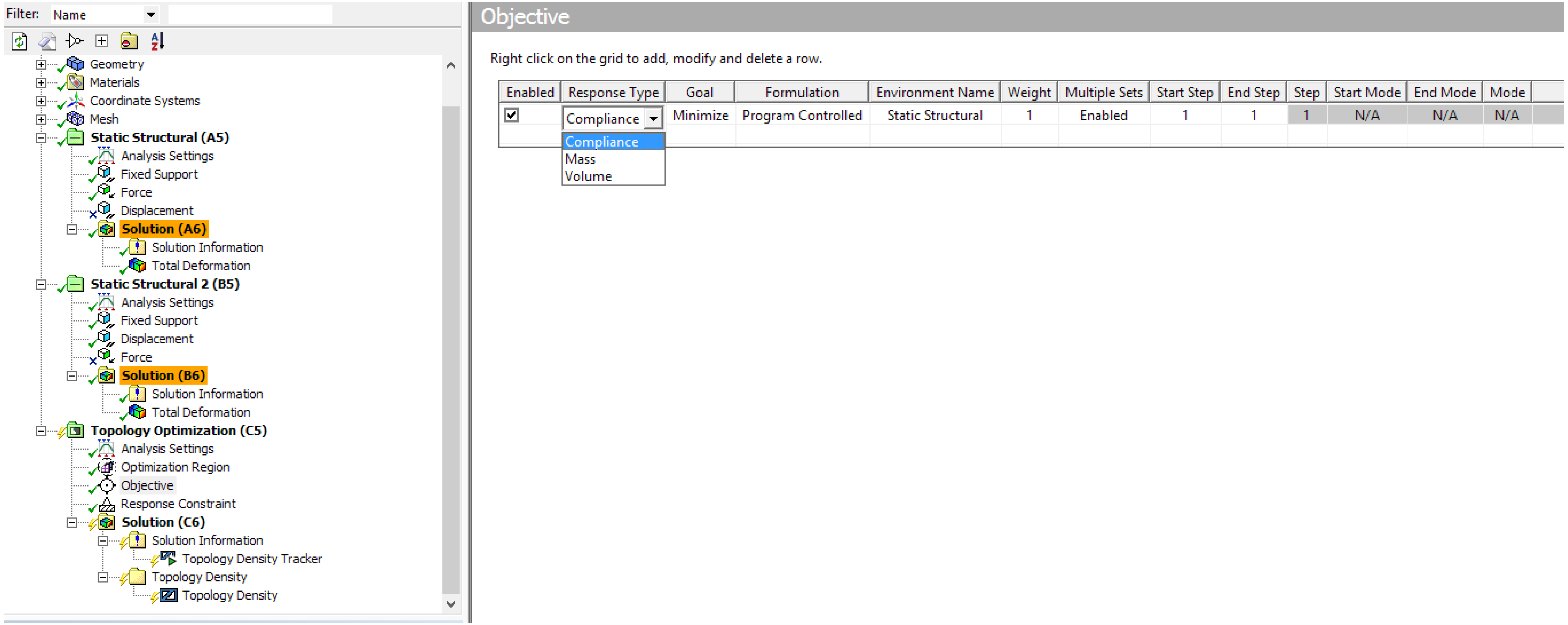
Level-set based Top Opt.

- Chose the « level-set solver »



Static Linear Analysis

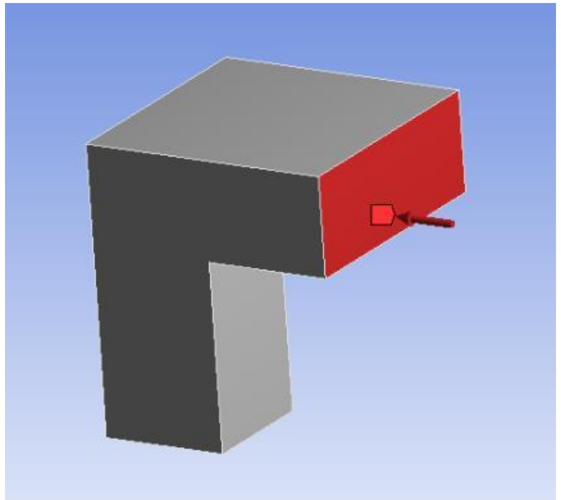
"Generalized" Compliance is supported as objective or constraint



Some examples

… without any smoothing !

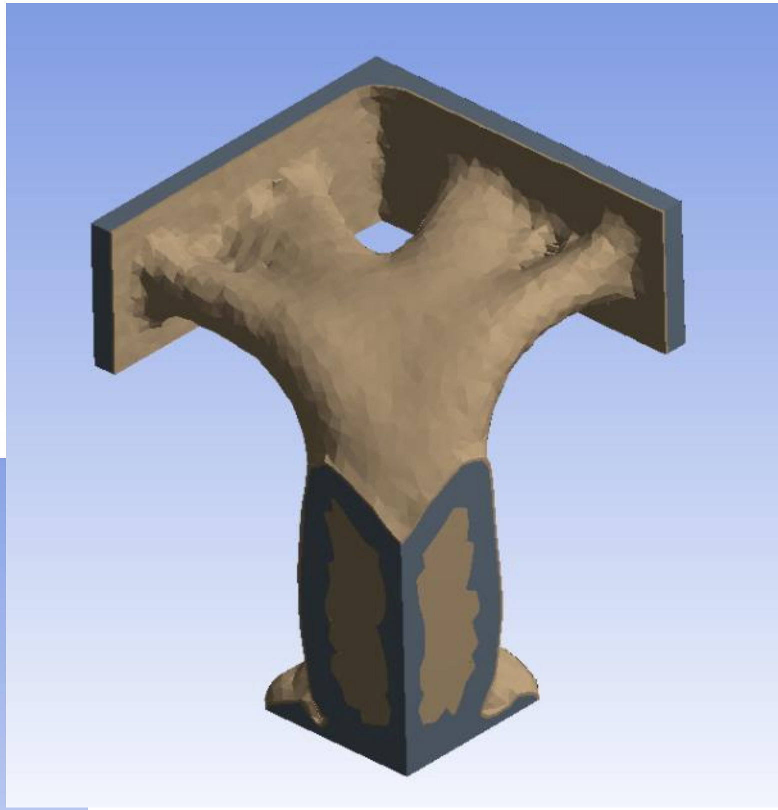
Use case: stopper



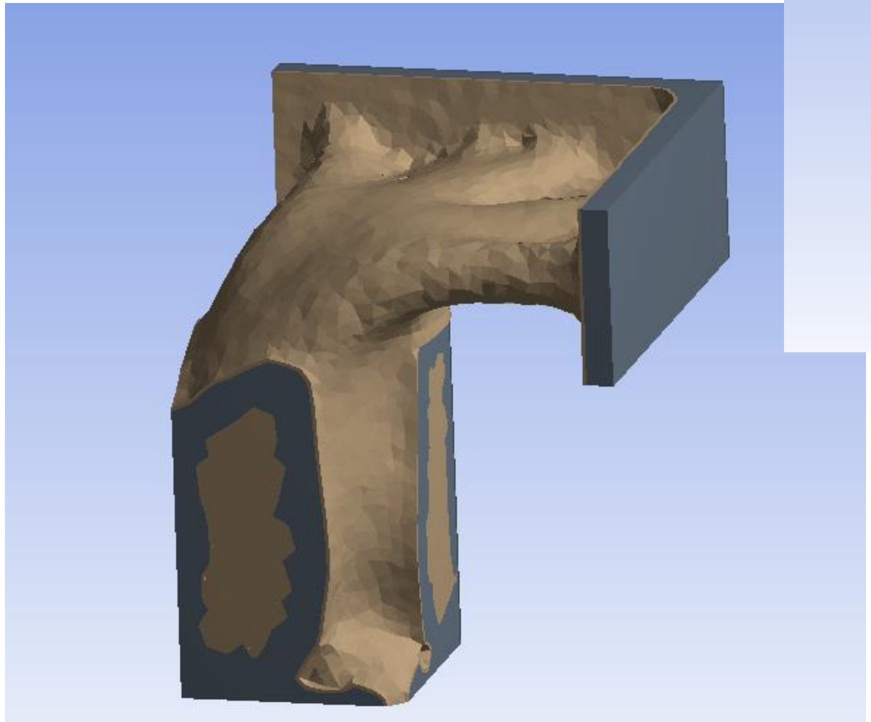
Mesh
- ✓ Tetra linear
- ✓ 117,000

Analysis
- ✓ Static lin (1): pressure

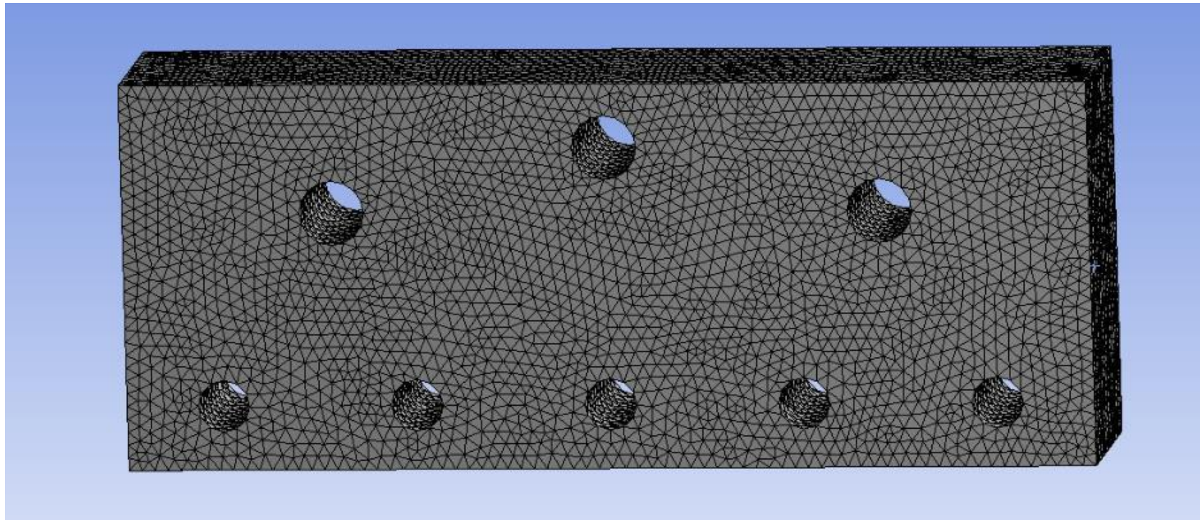
Optimization
- ✓ Min compliance
- ✓ St: mass<50%

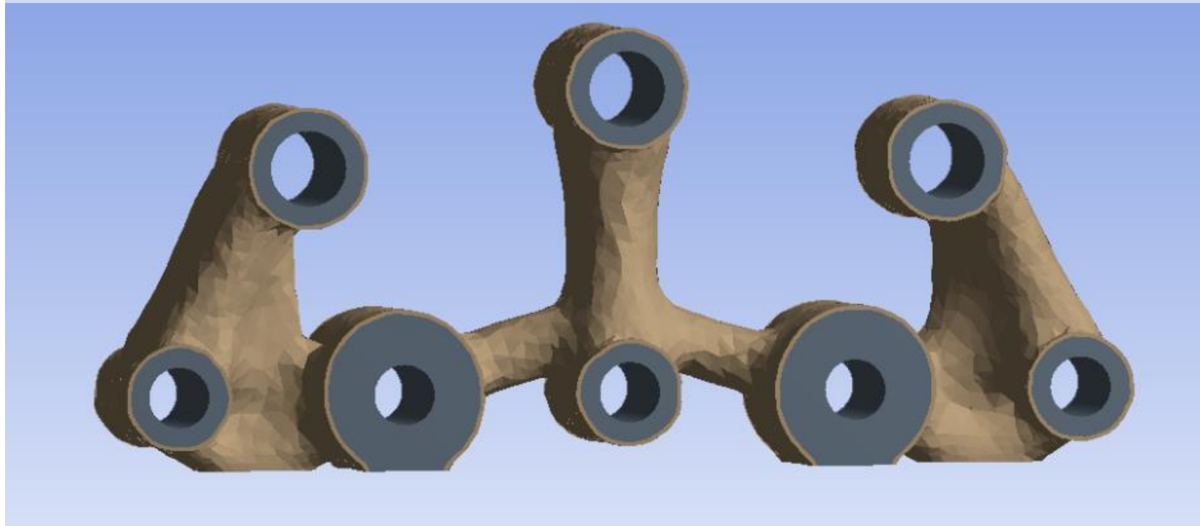


Result
- ✓ 27 iterations
- ✓ Current shape is feasible:
- ✓ (min) objective 0 : 7202.64
- ✓ constraint 0 : 1.84326e+07 <= 1.84475e+07



Use case:TOPO_OPT_LONG_WB2_002





Mesh

- ✓ Tetra linear
- ✓ 373,000

Analysis

- ✓ Static lin (1): force + pressure + remote force
- ✓ + nodal force + nodal pressure + remote disp +

Optimization

- ✓ Min comliance(1)
- ✓ St: mass<25%

Result

- ✓ 373kel : 52 iterations, objective1: 5.04038e-06, constraint 1: 3.89791, limit: 3.89848

Additive Manufacturing

ANSYS 2019R1 update

Summary

Workbench Additive

- Ability to orient part after geometry attach
- Layered tetrahedron meshing
- Heat treat step added
- Support restarts (add step after complete of build and cooldown)
- Allow powder in build step
- Allow non-build elements in build step
- Allow symmetry to be used
- Added 17-4 PH and AlSi10Mg to the sample materials
- Tmelt can be specified as a tabular entry (beta)
- Blade interference prediction (beta)

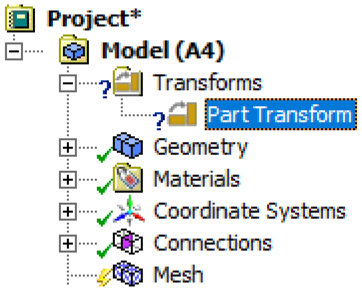
ANSYS Additive (Print & Science)

- New Voxelization Method
- User Defined Supports
- Single Bead Parametric (Additive Science)
- Porosity Parametric (Additive Science)
- Beta Thermal History (Additive Science)
- Thermal Solver Updates
- Thermal Mesh Control
- Updated Part Size Limitations
- Added Al357 and Ti64 Thermal
- Added Mesh Visualization Output

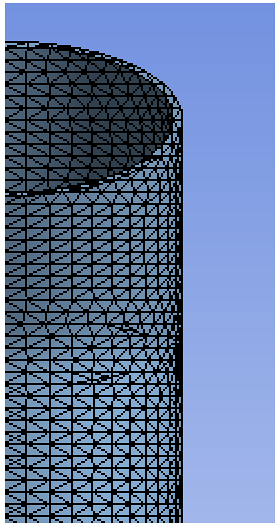
Ability To Orient Part After Geometry Attach

You can orient the part after importing it

It can be parameterized, so you can investigate the effect of orientation on support needs and on build distortion and stresses



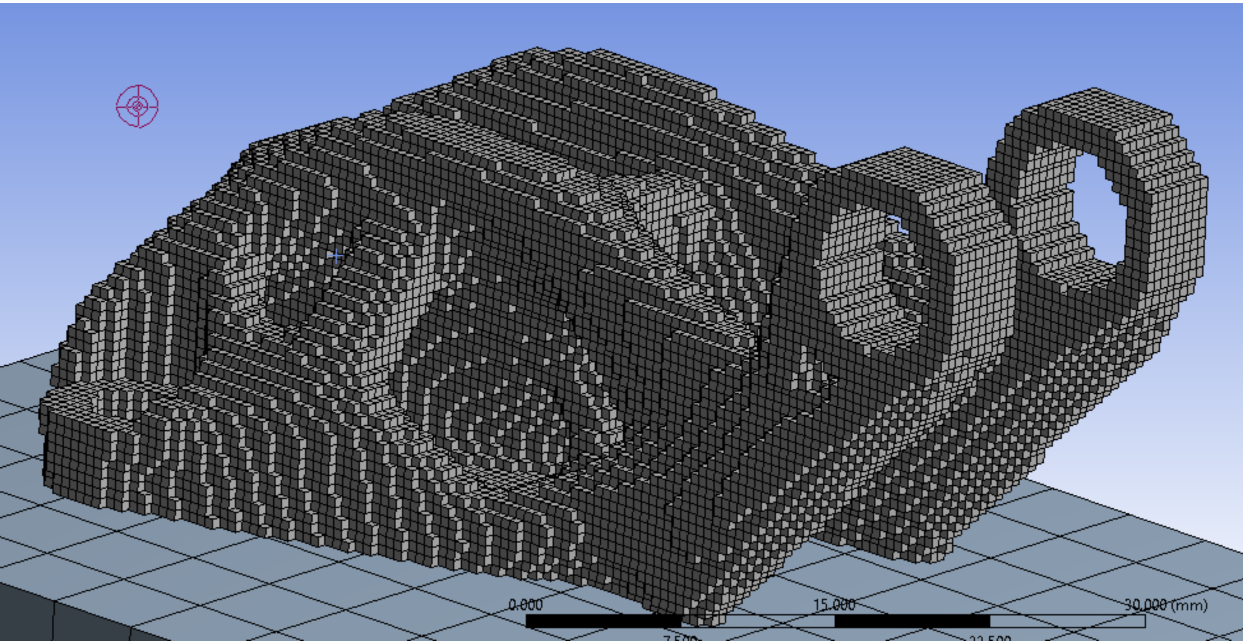
Layered Tetrahedron Meshing



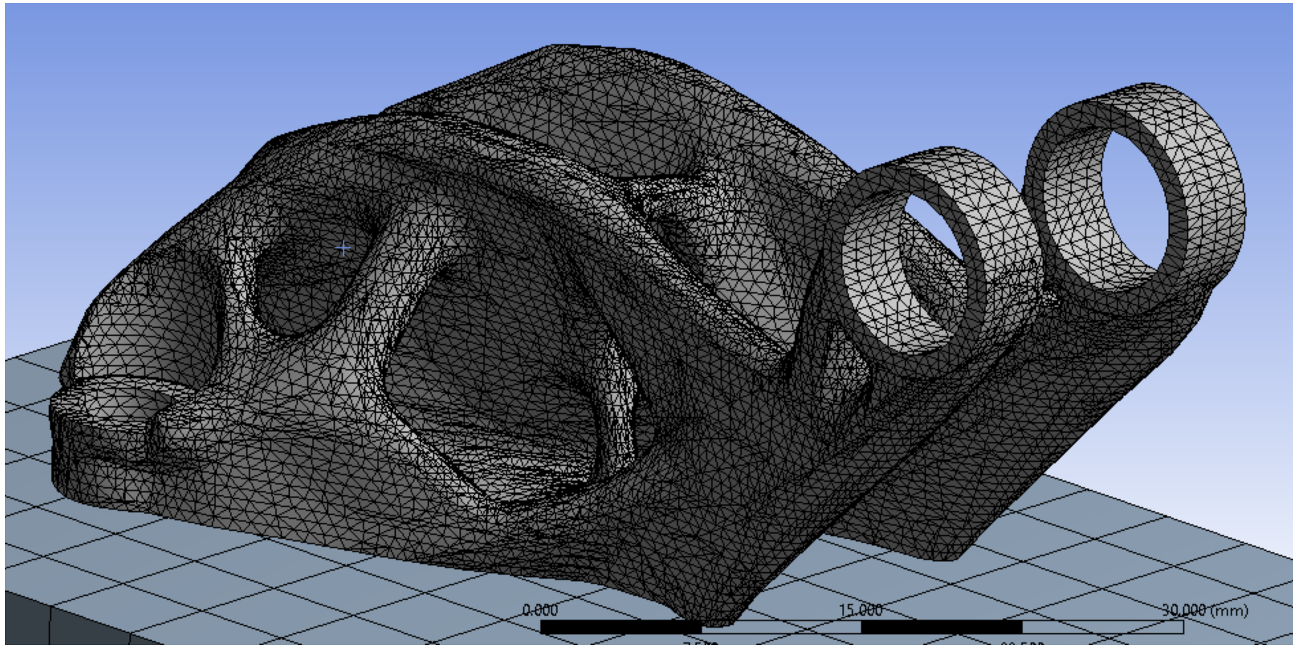
You can use a tetrahedral mesh for build simulation

- Captures geometry details
- More suitable for thin-walled parts
- Produces layered mesh

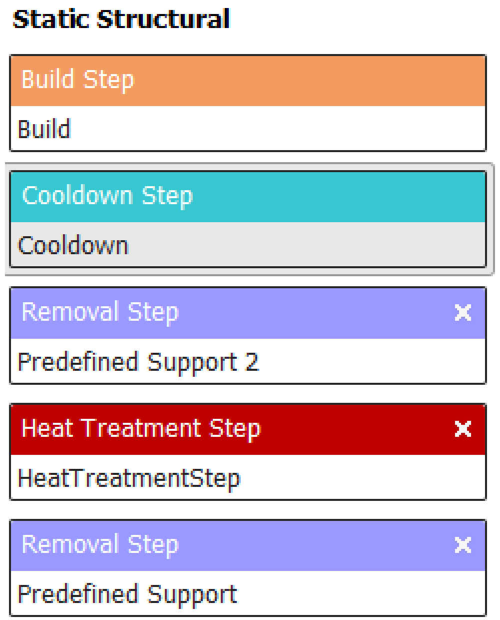
Cartesian Mesh



Layered Tetrahedron Mesh

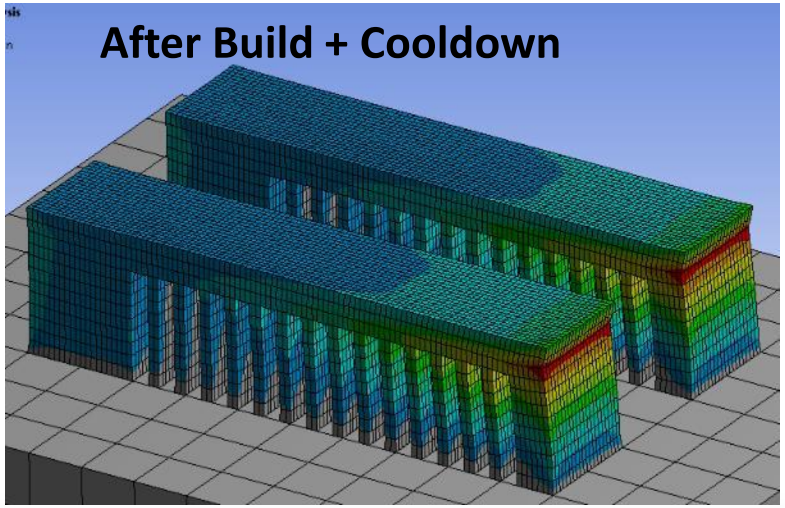


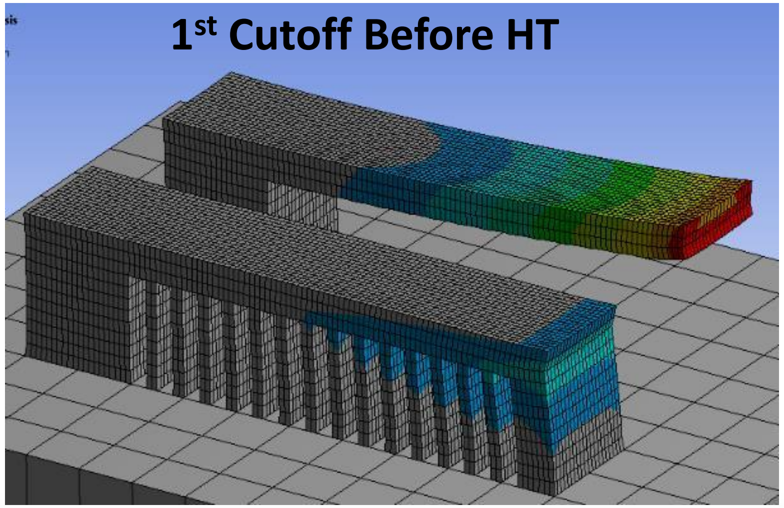
Heat Treat Step Added

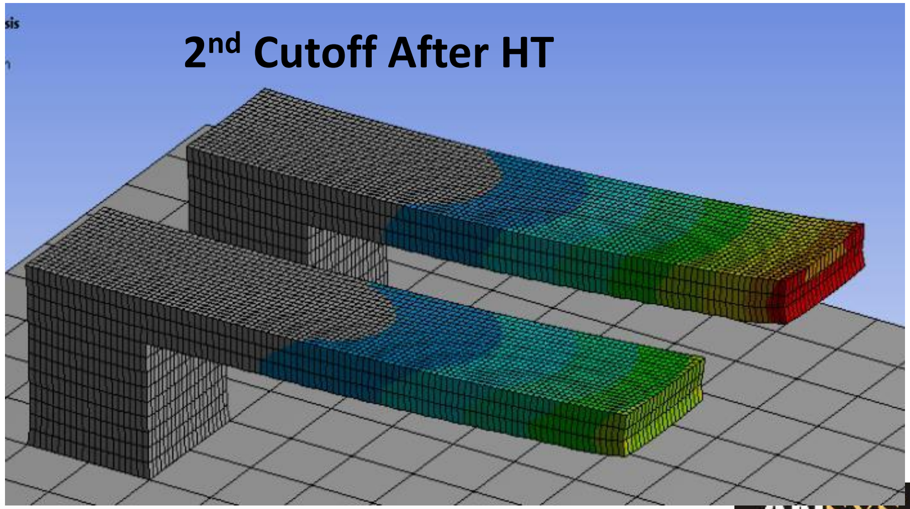


Can add a heat treat process step to stress relieve the part after it has been built

- **Stress relief before the part has been cut off or after**
- **Can use creep properties to predict the stress relief, or a simplified approach is available:**
  - If the temperature at a point during the heat treat exceeds a user-provided relaxation temperature, all the plastic strain history is removed (the material point annealed)







Support Restarts

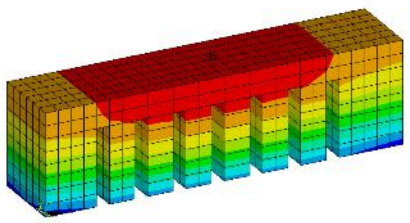
You can now add a step after the completion of the build and cooldown steps

- **Allows you to ensure those steps are properly performed before going on**
- **Also allows you to try different heat treat and/or removal steps**

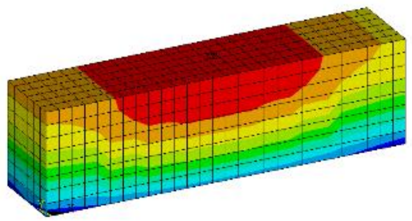
Allow Powder In Build Step

You can now include material that remains as powder during the build

- **Useful if you have multiple parts in close proximity on the build plate and the heat transfer between them is important to consider**
- **Also useful if you have a part with features close together where the heat transfer between them is important**



No Powder (Equivalent Convection)



With Powder

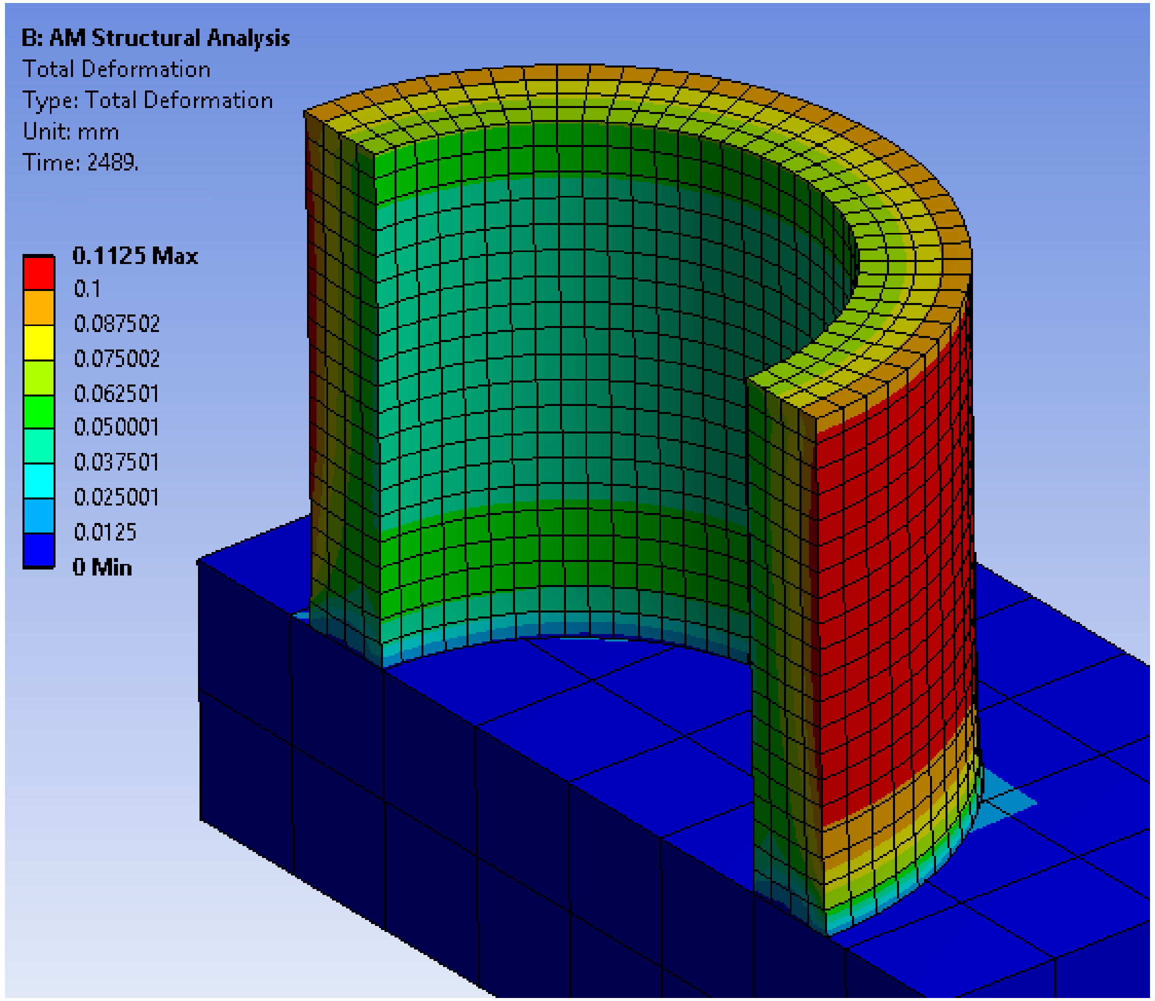
Allow Non-build Elements In Build Step

Experimental builds often have clamps, thermocouples, and other measuring devices inside the build chamber

You can now include them in the simulation as they will not be part of the build process (but will carry heat and stress)

Allow Symmetry To Be Used

If you have a symmetric part, you can use symmetry to reduce the model size



Added 17-4 PH And AlSi10Mg To The Sample Materials

17-4 PH steel and AlSi10Mg have been added to the list of available and validated materials

- **Both temperature-dependent thermal and structural properties are provided**

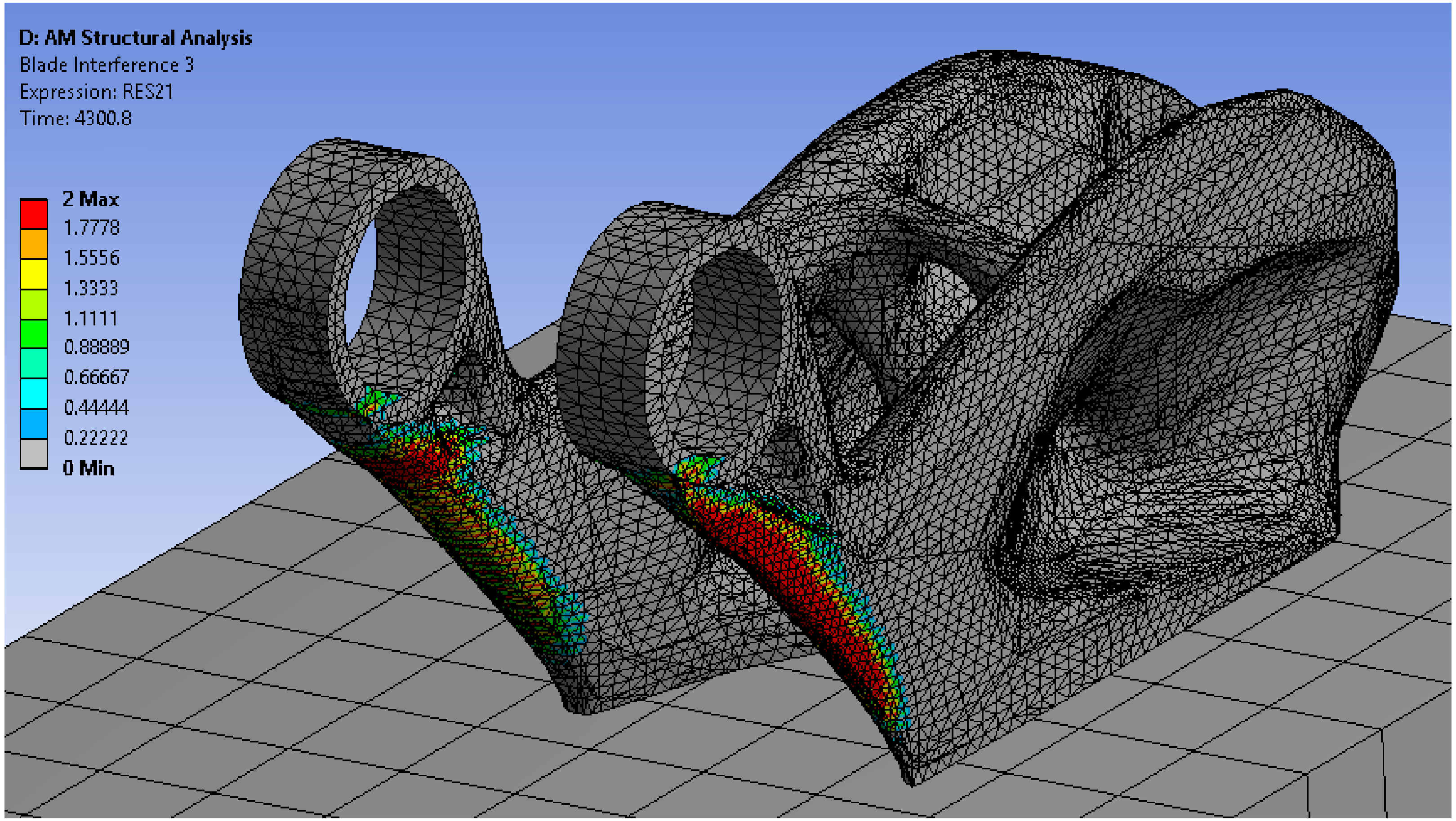
Tmelt Can Be Specified as a Tabular Entry (Beta)

The melt temperature can be specified as a tabular parameter

- **Tmelt can be a function of (x,y,z) or time**
- **Allows control of thermal history that some users are doing with the machine build settings**

Blade Interference Prediction (Beta)

You can predict areas where there is likely blade interference (blade crash) during the build



New Voxelization (AP)

Subdivide voxels in pre-processing to get voxel density approximation

Helps to address meshing problems with extremely fine features on large components

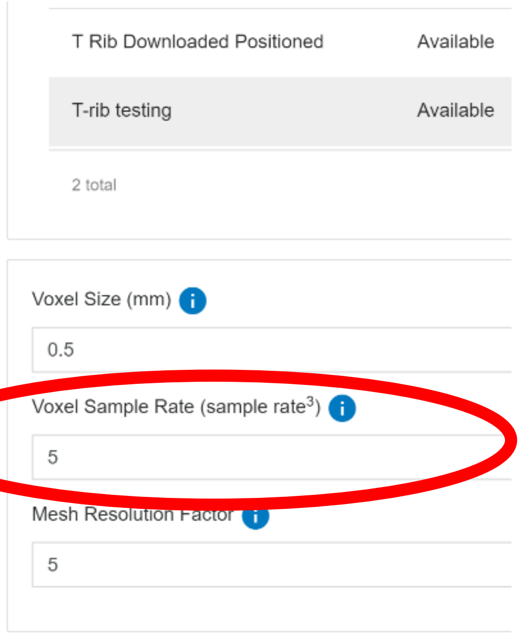
Voxel Sample Rate

Users can view voxel density file

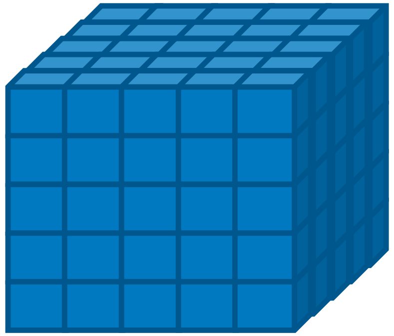
Mechanical properties are scaled based on voxel density

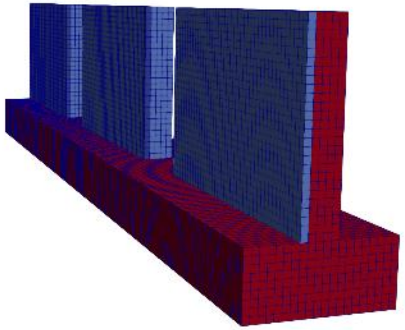
More accurate distortion

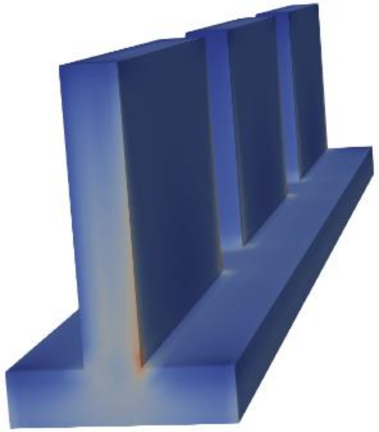
Limitation: stress visualizations are known to look odd due to stiffness of partial density voxels

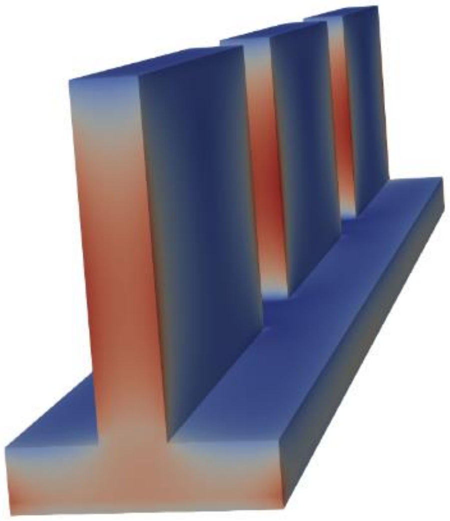


Sample Rate of 5
= 125 Subvoxels









Uploading Supports (AP)

Utilizes new voxelization

User can design their support structures

User can upload multiple support designs per part

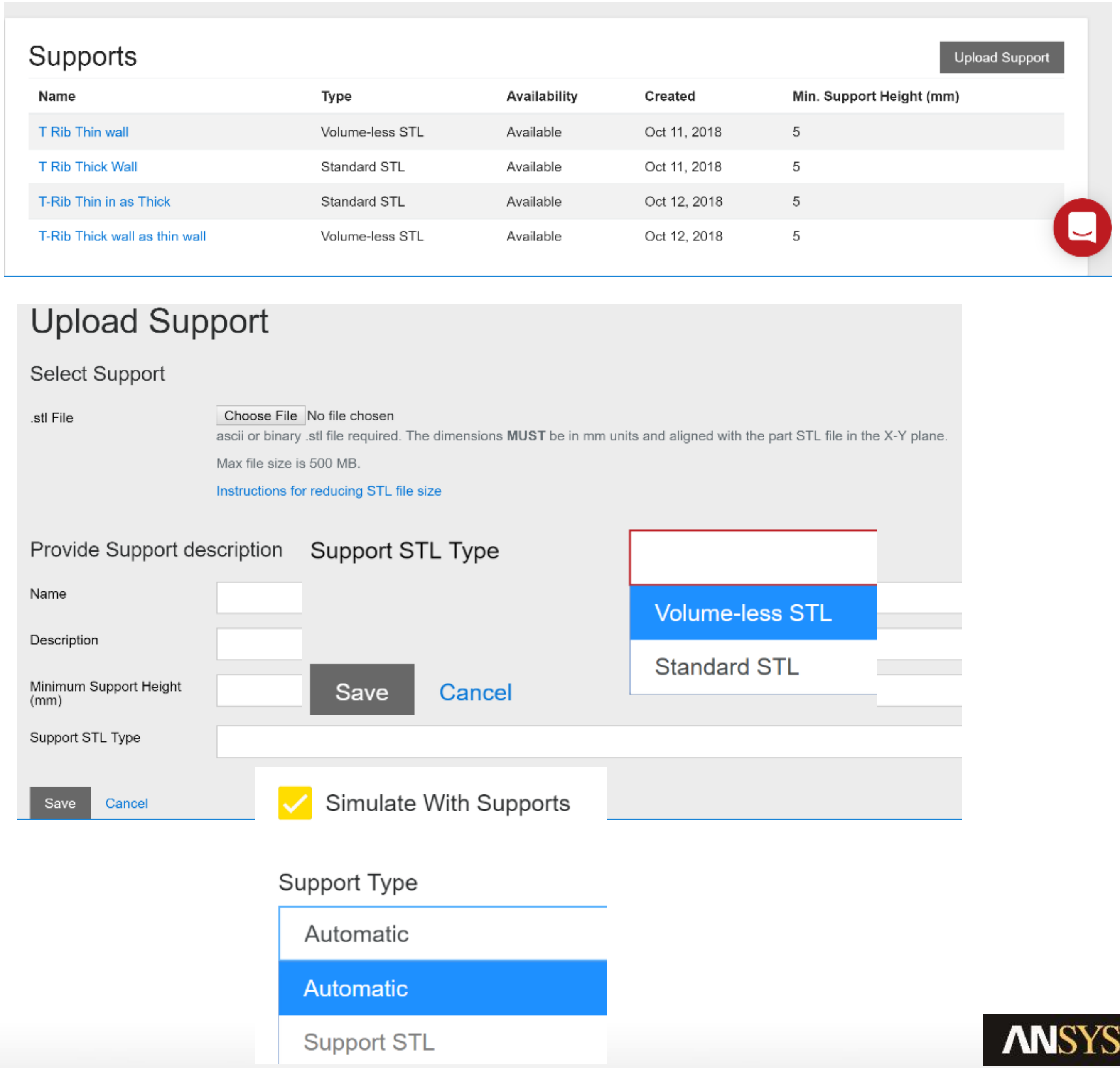
Approximate geometry and supplement with voxel density

More accurate comparisons on customer parts

Addresses the request of many customers during evaluation period

Standard or Volumeless support .stl files

Limitations: Cannot output optimized supports



Single Bead Parametric (AS)

New Additive Science Feature

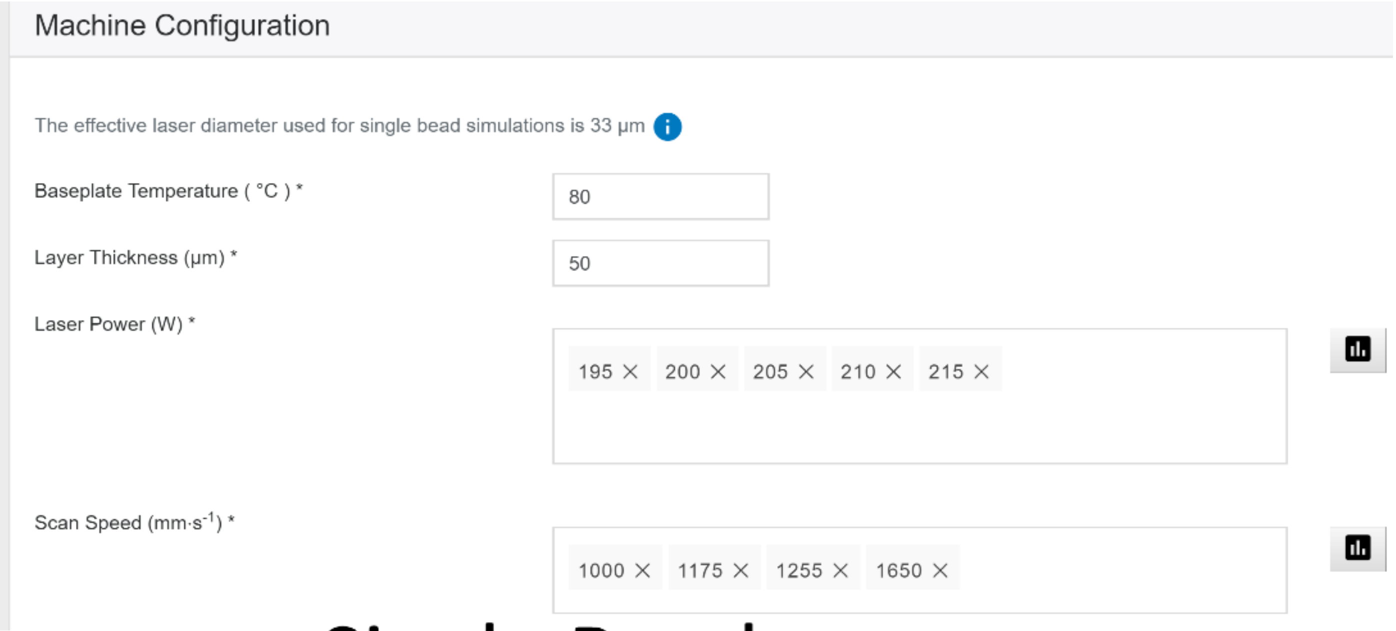
Dimensions of meltpool (Length, Width, Depth)

Helps users set machine parameters of power and speed

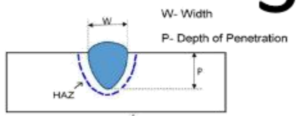
Parametric Capability: up to 300 permutations

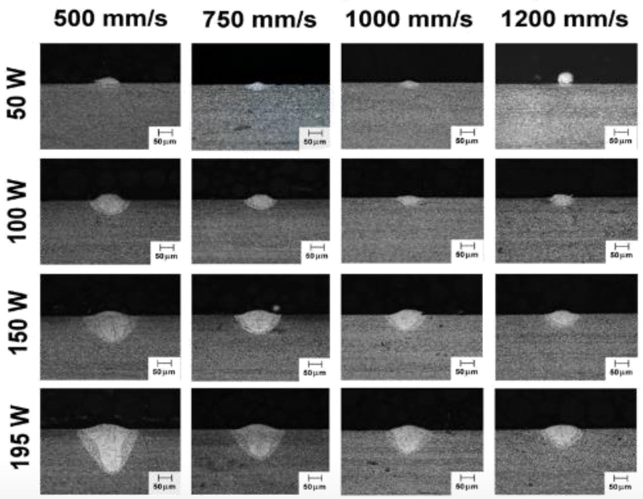
Tuned and validated for multiple materials (IN718, IN625, CoCr, 17-4 PH, Ti64, Al357)

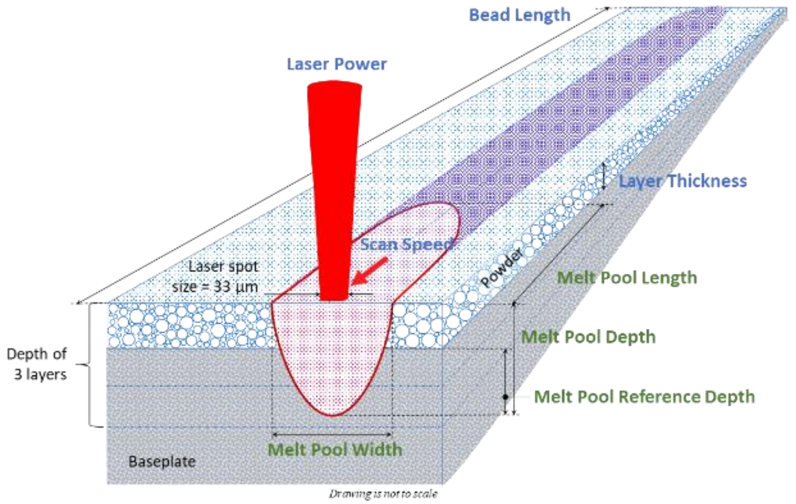
Limitations: Tuning function only supports a specific range of parameters, so range is limited in UI.



Single Bead







Single bead scan on powder showing melt pool dimensions
(Inputs in blue, outputs in green)

Porosity Parametric (AS)

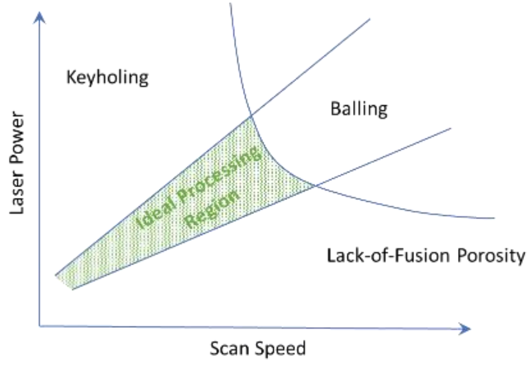
New Additive Science Feature

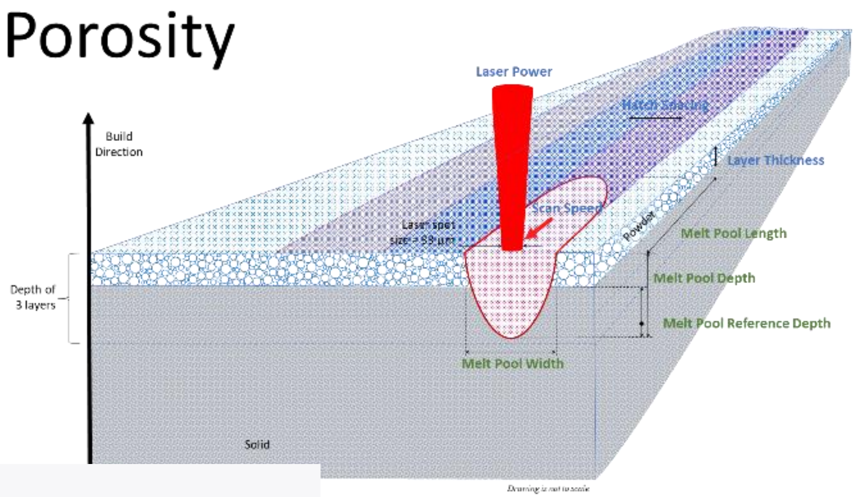
Helps users set process parameters of scanning strategy in addition to power and speed

Parametric Capability: up to 300 permutations

Tuned and validated for multiple materials

Limitations: Tuning function only supports a range of parameters, so range is limited in UI.





Machine Configuration

Machine: Generic
Selecting a new machine will override the machine properties with machine defaults

Baseplate Temperature (°C) *: 80

Starting Layer Angle (°) *: 57

Layer Rotation Angle (°) *: 67

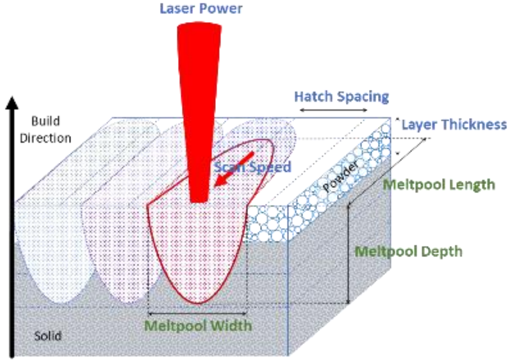
Laser Power (W) *: 195 ×

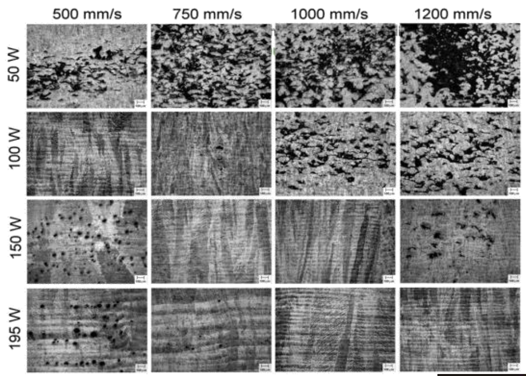
Scan Speed (mm·s⁻¹) *: 1000 ×

Layer Thickness (µm) *: 50 ×

Hatch Spacing (mm) *: 0.1 ×

Slicing Stripe Width (mm) *: 10 ×





BETA Thermal History (AS)

New Additive Science Beta Feature

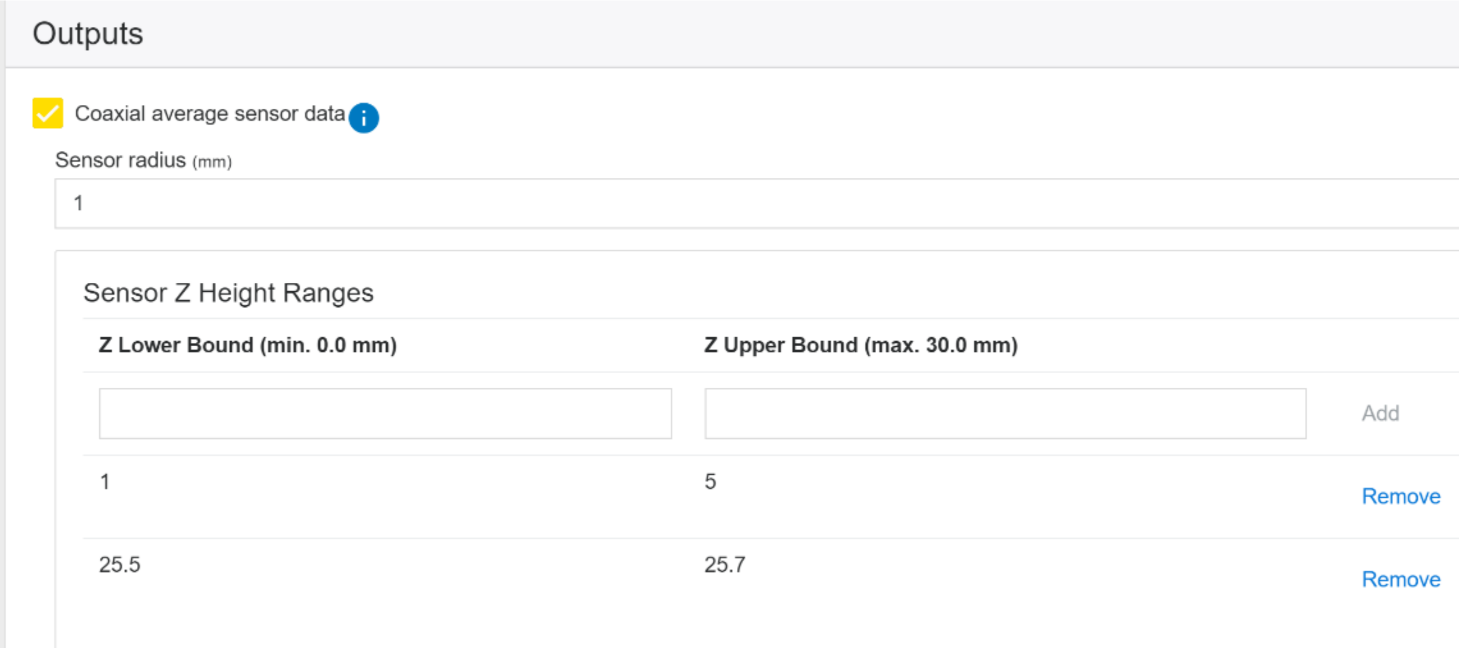
Helps users to visualize temperature profile of a part

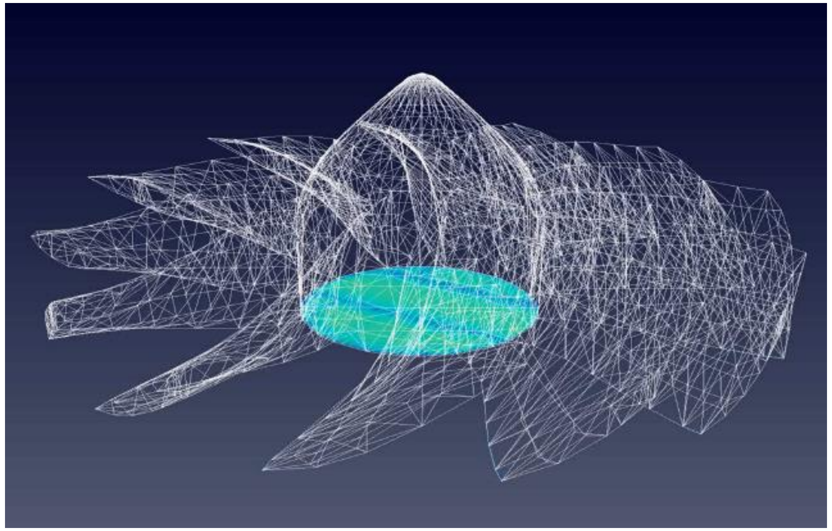
Allows users to evaluate meltpool dimensions based on scan vectors of a specific part

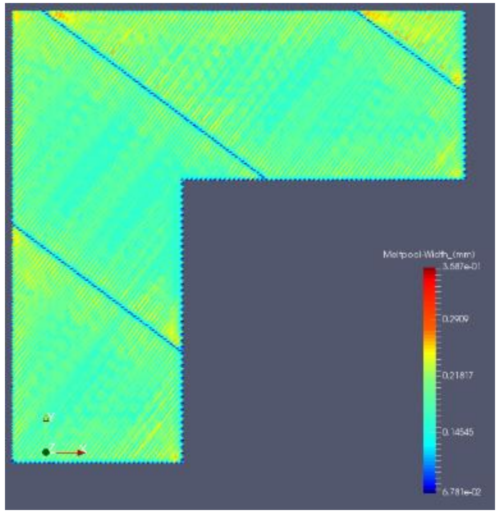
Users choose their designed geometry

User selects layers of interest

Coaxial average sensor allows users to choose radius of sensor over which to average data points

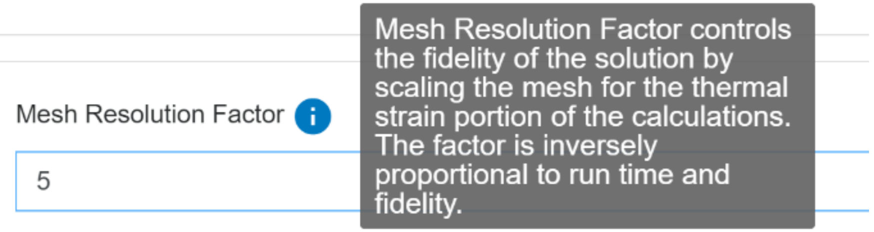


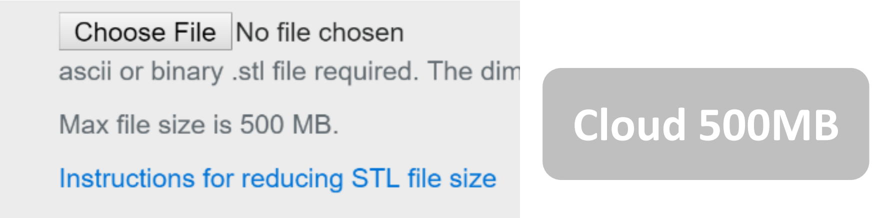


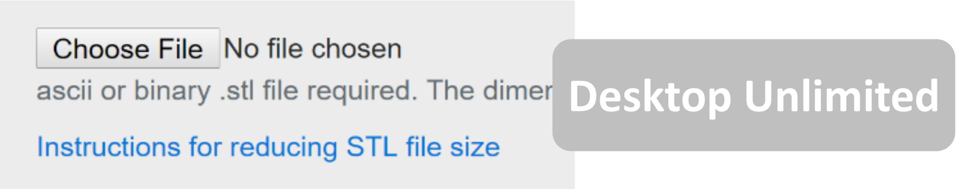


Existing Feature Updates (AP & AS)

- Thermal Solver: the thermal solver has undergone multiple updates to more accurately approximate the physics, which provides more realistically accurate thermal strain results
- MRF parameter introduced: This is a Mesh Resolution Factor, which controls how much the full fidelity thermal mesh will be expanded for thermal strain and thermal history simulations. This parameters allows users to control the speed, fidelity, and data generation for simulations
- Part Size limitations: removed for desktop increased to 500 MB for Cloud
- Download .stl component from an uploaded build file
- Expanded thermal simulation material database to include Ti64
- Added Al357 for thermal and mechanical simulations
- Support Parameters have been updated to be more clear and provide users with clarity in controlling optimized support structures that are output
- Added voxelized mesh file output, so that users can view the mesh as soon as the simulation starts instead of needing to wait for the entire simulation to complete before mesh can be visualized







Materials

Search...

Name	Key	Source	Date Modified	Available for Thermal Simulation
17-4PH	17-4ph	ANSYS Pre-defined	4/9/18, 10:04 AM	✓
Al357	Al357	ANSYS Pre-defined	11/12/18, 4:41 AM	✓
AlSi10Mg	AlSi10Mg	ANSYS Pre-defined	4/9/18, 10:04 AM	✕
CoCr	CoCr	ANSYS Pre-defined	4/9/18, 10:04 AM	✓
IN625	Inc625	ANSYS Pre-defined	4/9/18, 10:04 AM	✓
IN718	Inc718	ANSYS Pre-defined	4/9/18, 10:04 AM	✓
Ti64	Ti64	ANSYS Pre-defined	4/9/18, 10:04 AM	✓

7 total

Composites 2019 R1 Release Notes

ANSYS Composite PrePost (ACP)
ANSYS Composite Cure Simulation (ACCS)

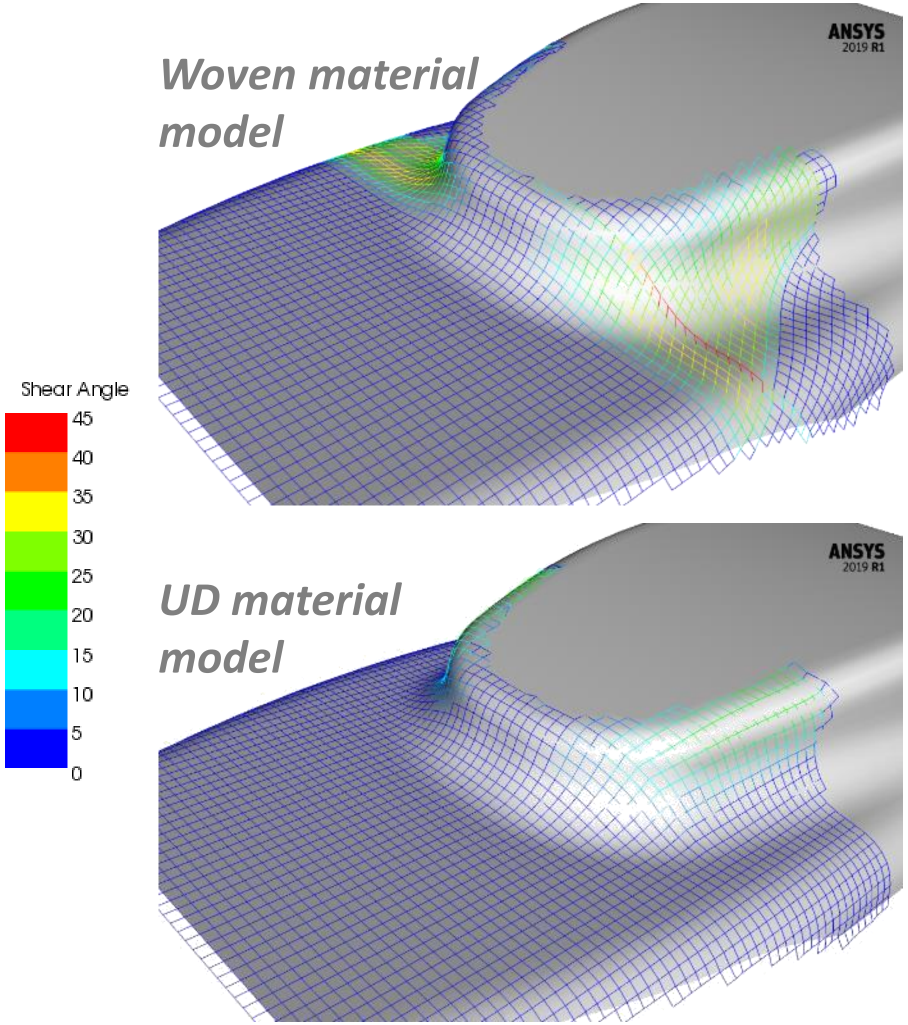
Outline

- Composite PrePost (ACP)
  - **UD Draping**
  - **Cut-off Feature for The Lay-up Mapping**
  - **Geometrical Selection Rule**
  - **Material Plot**
  - **New Serialization Format**
  - **Miscellaneous**
- Composite Cure Simulation (ACCS)

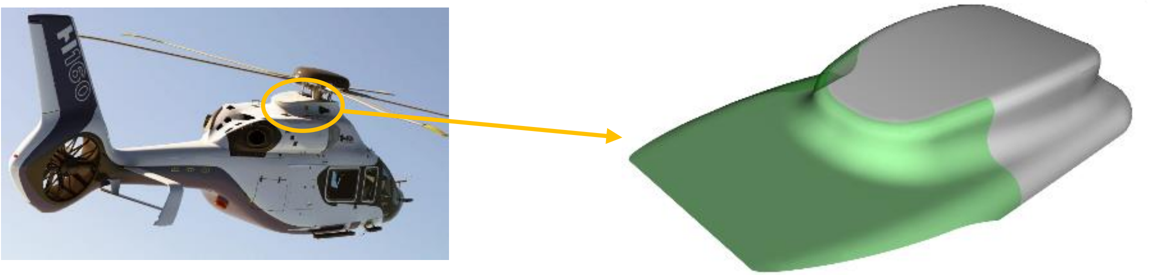
Draping

- The draping functionality has been extended to unidirectional (UD) fabrics. The draping algorithm now differentiates between a woven and a UD material model.

- The transverse deformation of the draping mesh (modeling spreading and compacting of fibers) can be controlled with the UD coefficient.

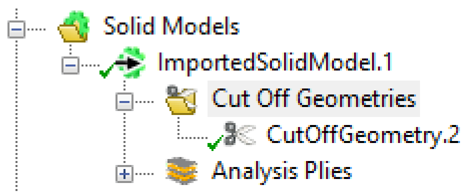


Effect of the draping material model on the shear angle calculated by ACP draping simulation.

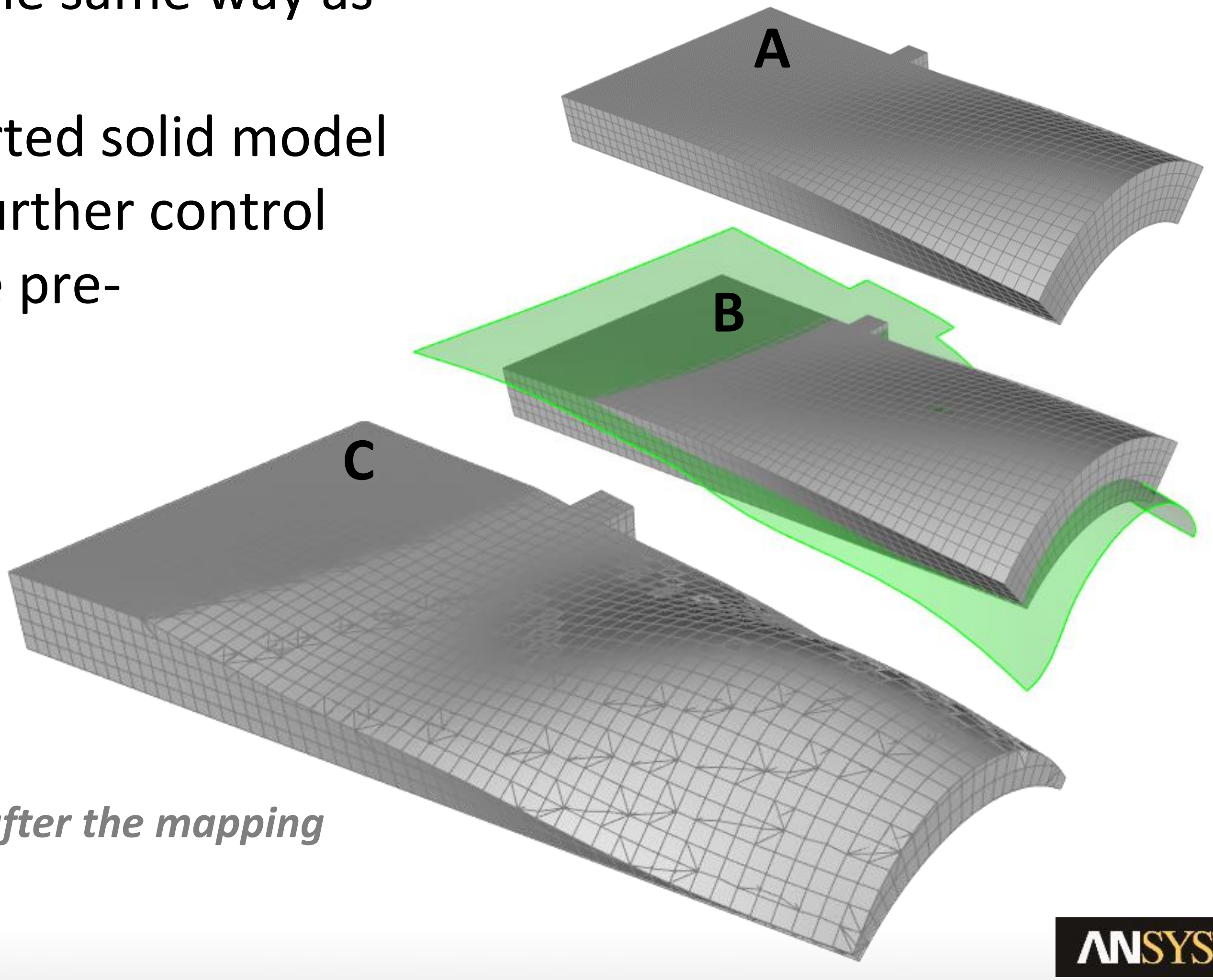


Draping a patch in the back of a helicopter rooftop.

Cut-off Geometry Feature for the Lay-up Mapping



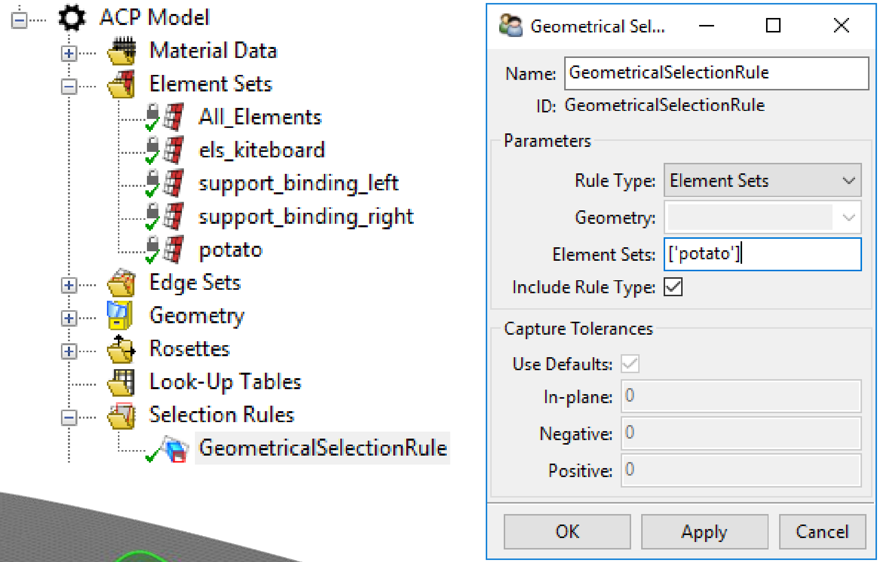
- The lay-up mapping feature for solid meshes now supports the cut-off geometry feature in the same way as the standard extruded solid model does.
- This feature allows you to shape the imported solid model after the lay-up mapping. That gives you further control of geometry and mesh and it simplifies the pre-processing.

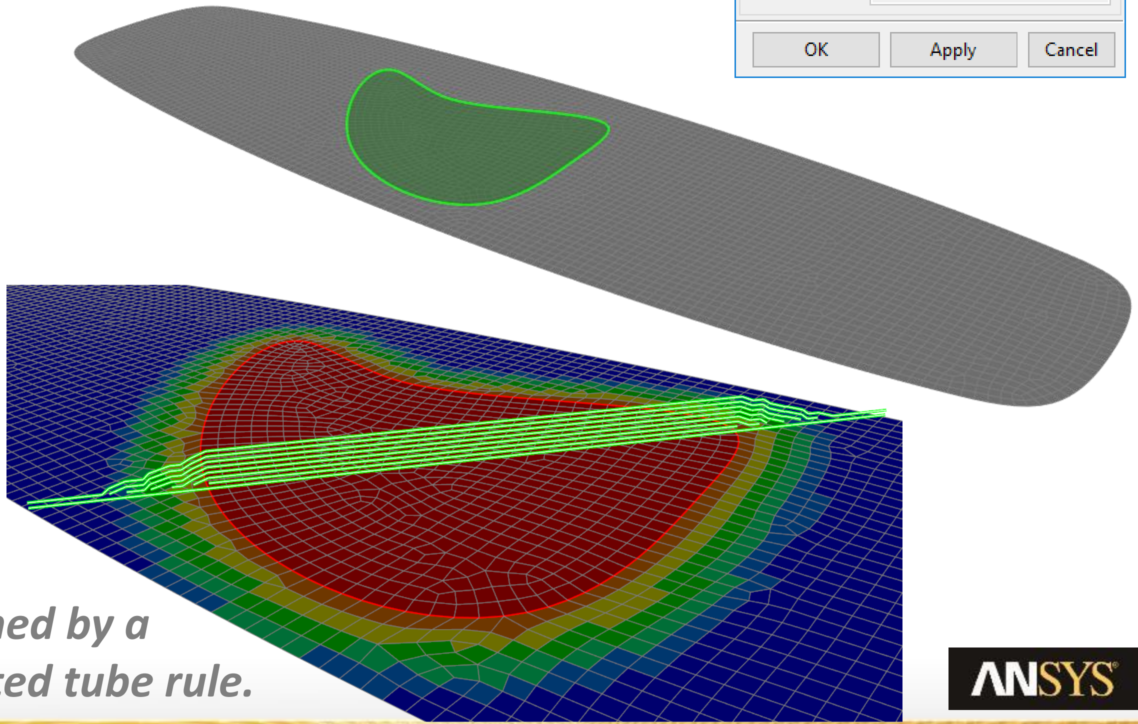


A) Simple solid with structured mesh
B) Cut-off geometry that is applied after the mapping
C) Shaped solid with mapped lay-up

CAD Selection Rule becomes Geometrical Selection Rule

- The Geometrical Selection Rule allows for a parametrized lay-up definition in relation to Element Sets. As a result, you can now easily define a ply staggering from an Element Set inward or outward.
- The functionality of the former CAD selection rule remains the same.
- The Geometrical Selection Rule can thus be based on CAD geometries as well as Element Sets by switching the rule type.



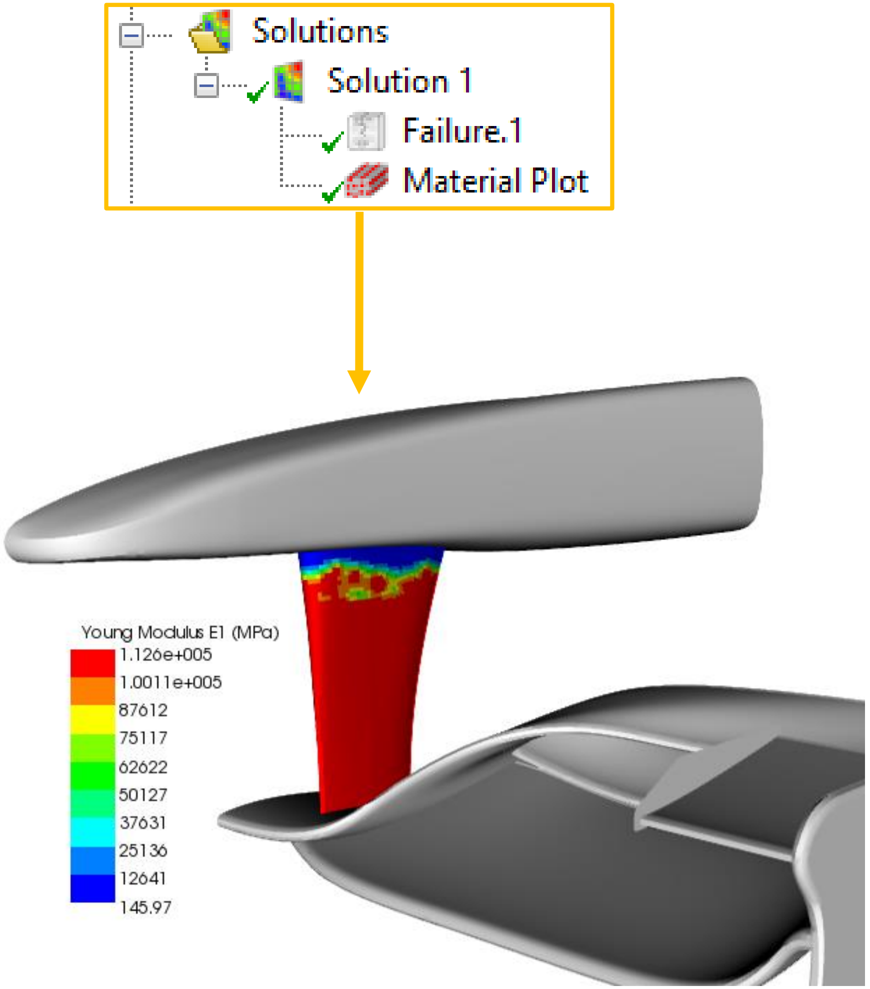


Outward ply staggering defined by a geometrical rule and templated tube rule.

ANSYS

Material Plot

- The new Material Plot allows to plot variable material properties such as orthotropic elasticity, density, strain and stress limits at a ply-wise level.
- It is available as lay-up and solution plot. The latter one also allows to consider temperature dependency.
- Reviewing the effect of field variables (shear angle, temperature and user-defined variables) on the mechanical properties of the material has been greatly enhanced.



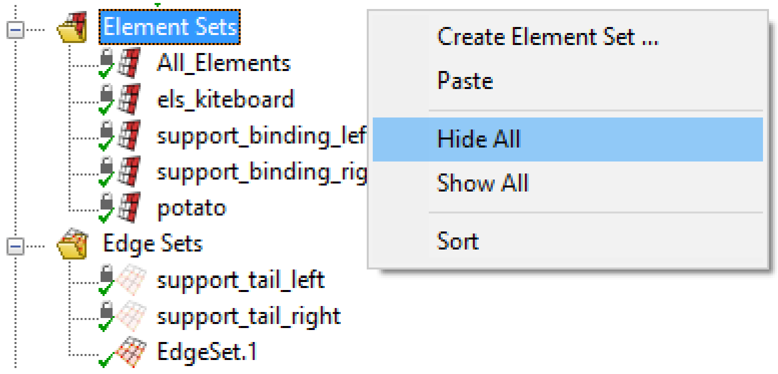
Plot of the Young's Modulus in the fiber direction as a function of Temperature, Shear Angle and Curing status.

New Serialization Format ACPH5

- The serialization has moved from a text to the binary ACPH5 serialization format.
- The composite workflows in WB are not affected by this change and previous models can be upgraded to the new format with ease.
- The new format is more efficient and flexible.

Miscellaneous

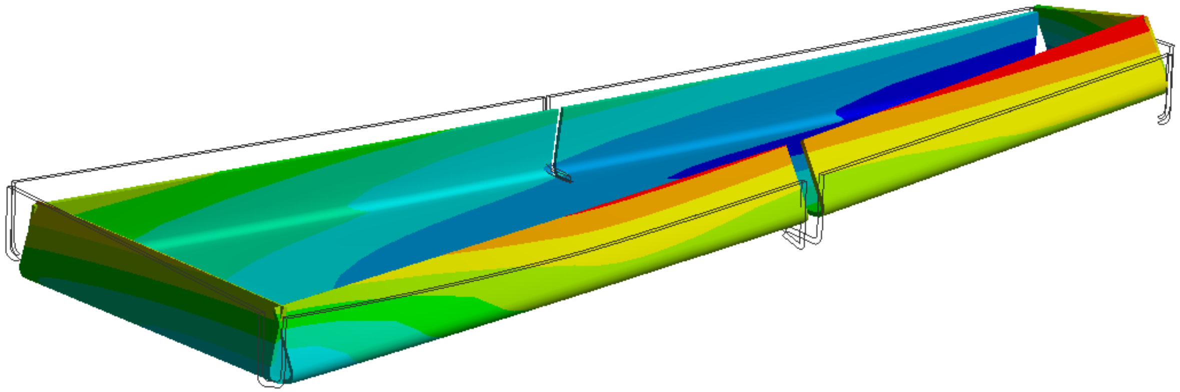
- GUI Improvements
  - **Hide and Show All actions have been added to some groups such as Element Sets, Edge Sets etc. to control the visualization for all items in a group**
  - **For direction definitions a flip button is now available enhancing the usability.**
- The performance bottle neck when loading models with thousands of plies has been resolved.
- Various bug fixes.



ANSYS Composite Cure Simulation (ACCS)

Improvements in the Cure Simulation workflows and ACCS module:

- ACCS supports now variable material properties during the cool down (solidified material state).
- Bugs in the unit handling have been resolved.
- The stability, especially for bigger models, has been improved.

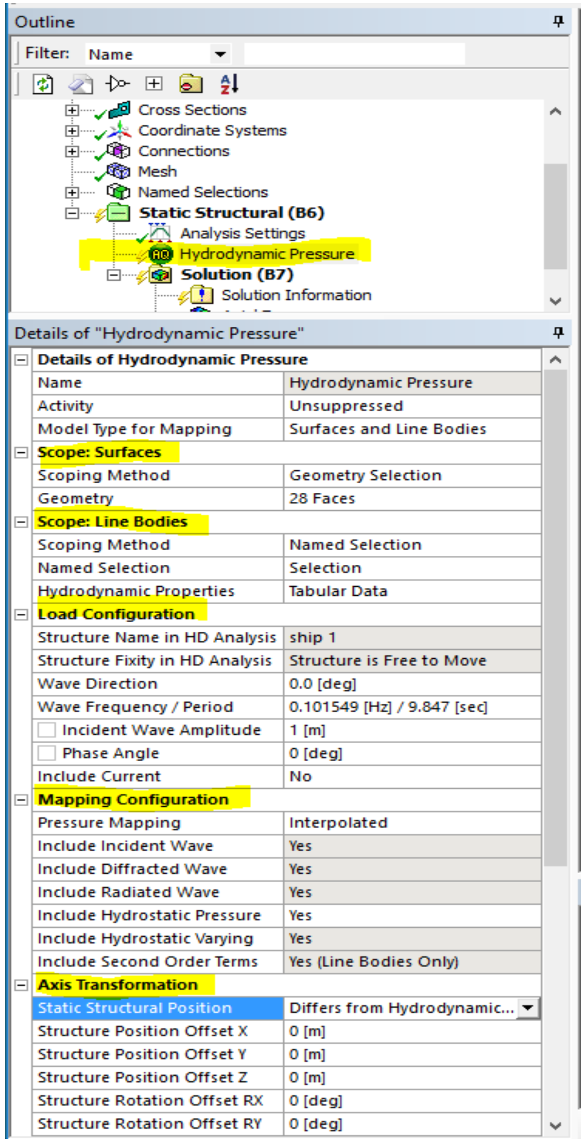


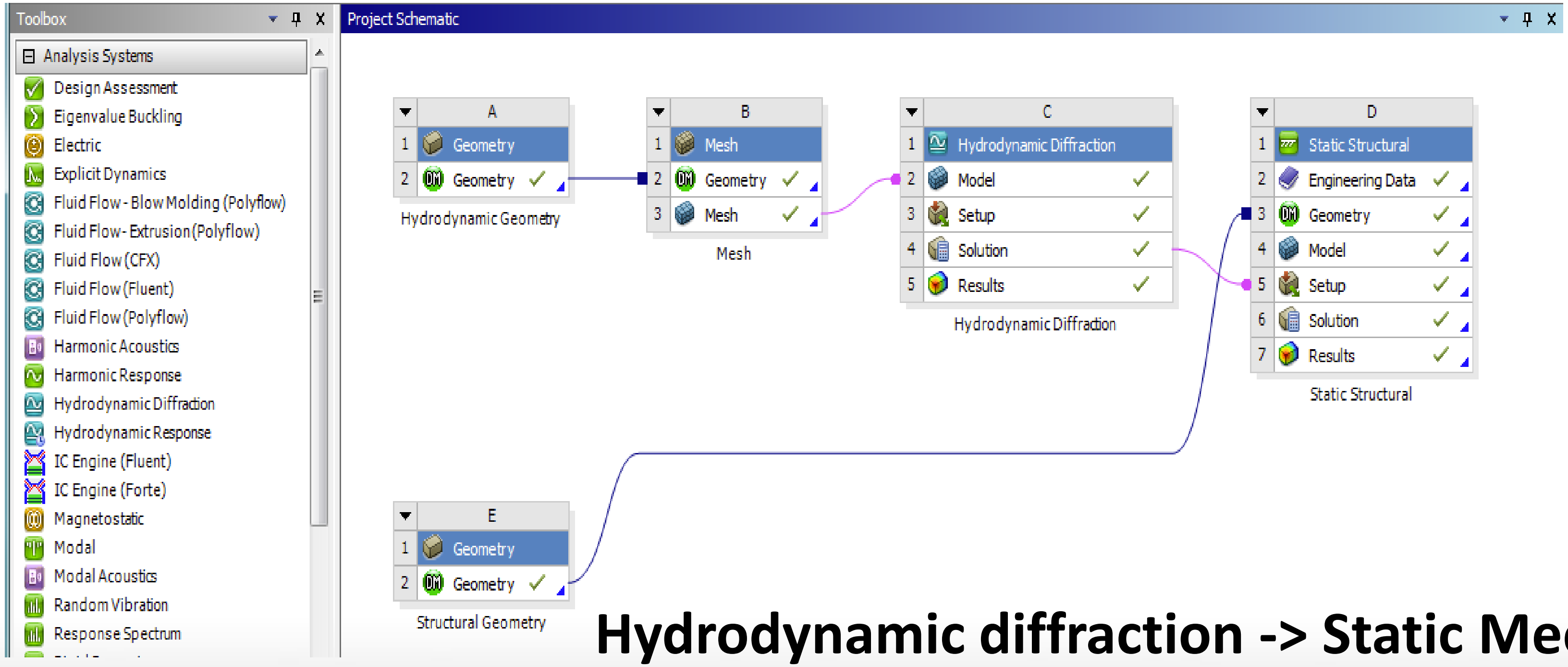
Mechanical Aqwa

ANSYS 2019 R1 update

SA799: Hydrodynamic load mapping ACT extension

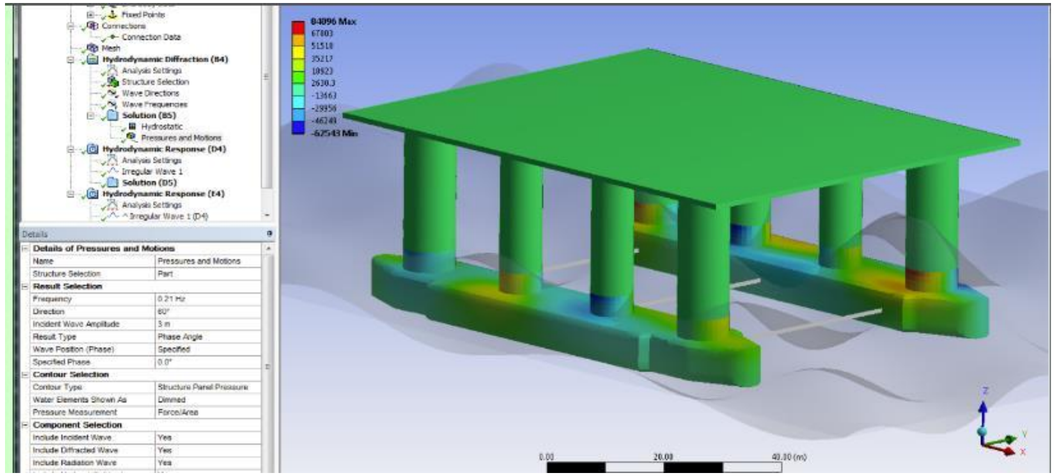
- Replaces previous multi-step workflow
- Simplifies the load transfer process significantly
- Represent results in WB
- Valid for combination of solid/surface and line elements



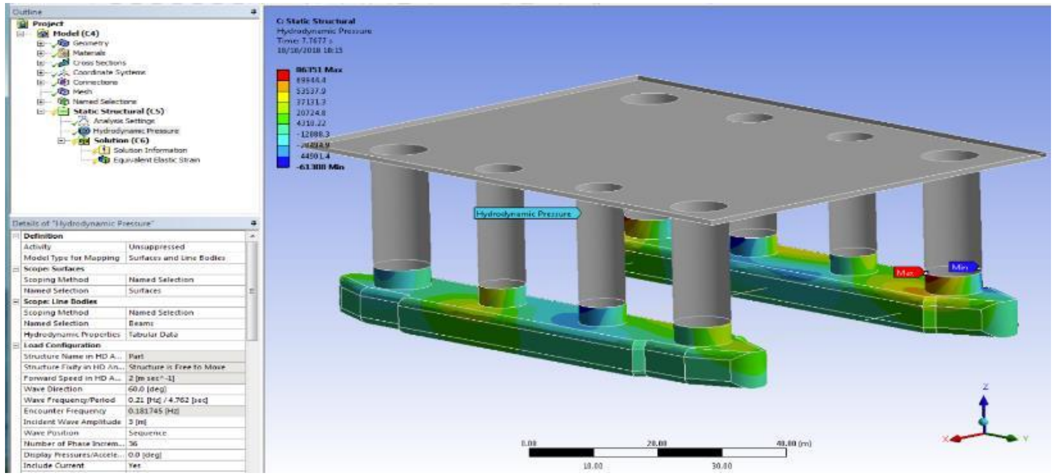


Hydrodynamic diffraction -> Static Mechanical

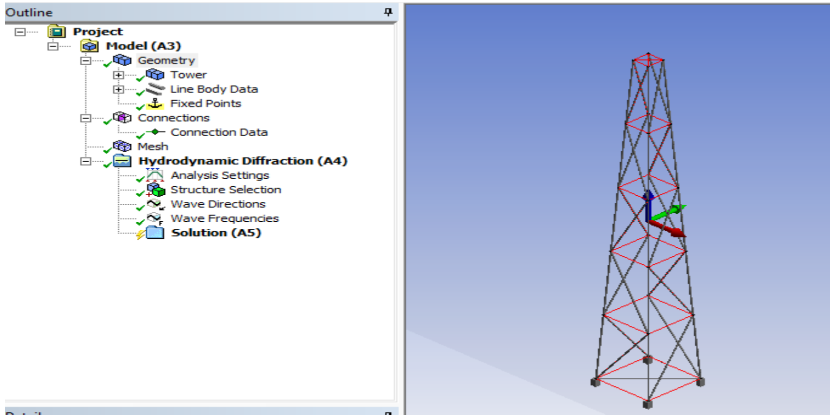
SA799: Hydrodynamic load mapping ACT extension



Pressure/loads on hydrodynamic model

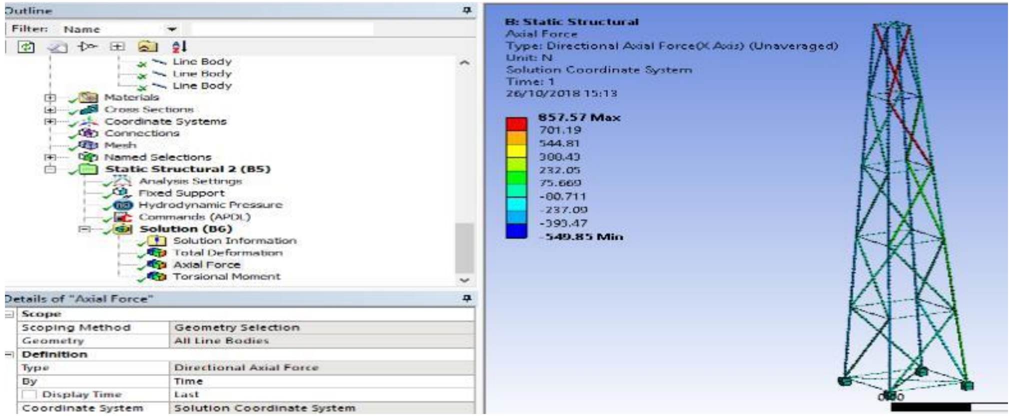


Mapped pressure/loads on FE model



HD model of fixed jacket platform

(Morison equation for load on tube)

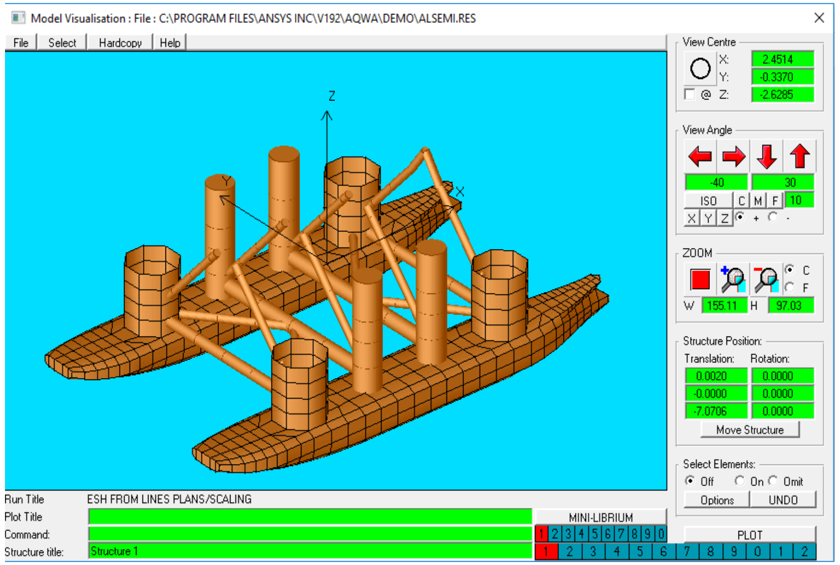


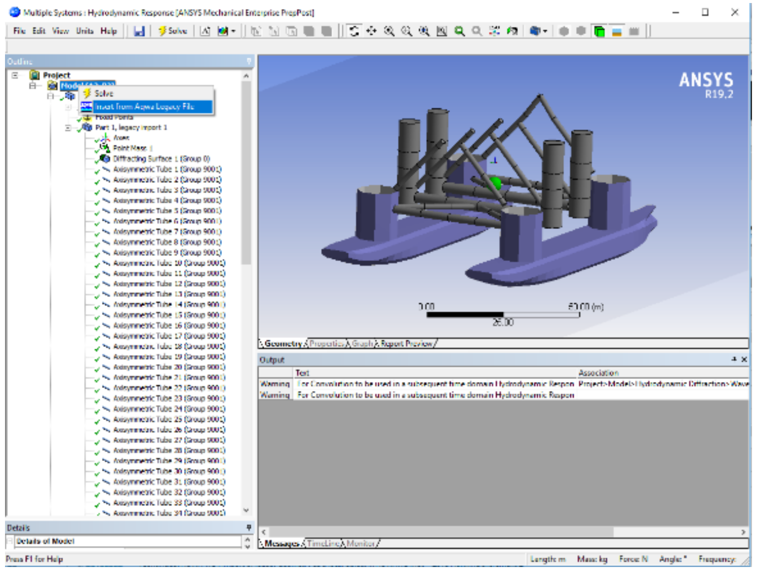
Axial force on FE model

ANSYS

SA800: Aqwa model data file import

- To encourage existing users with old models to move to WB
- To transfer data from one project to another.

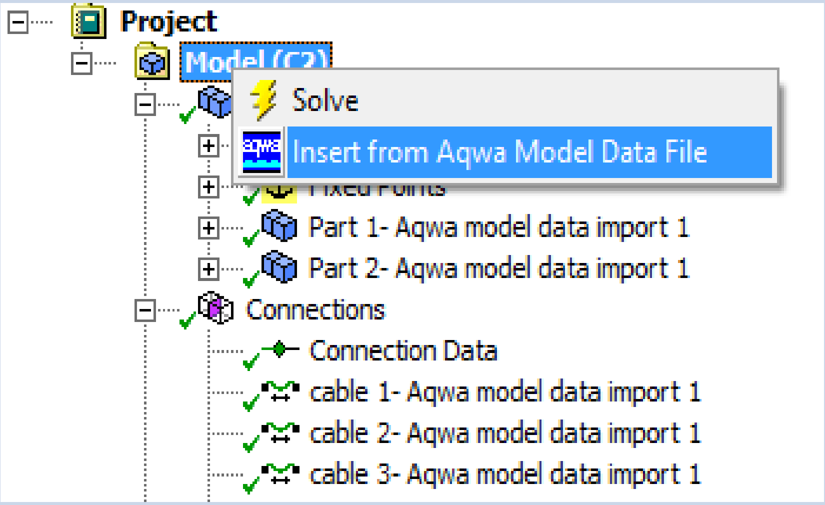


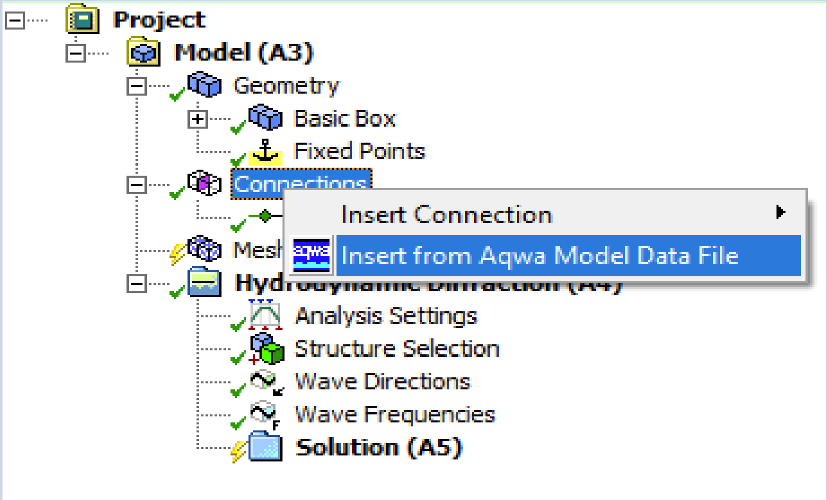


SA800: Aqwa model data file import

- Flexible import to WB

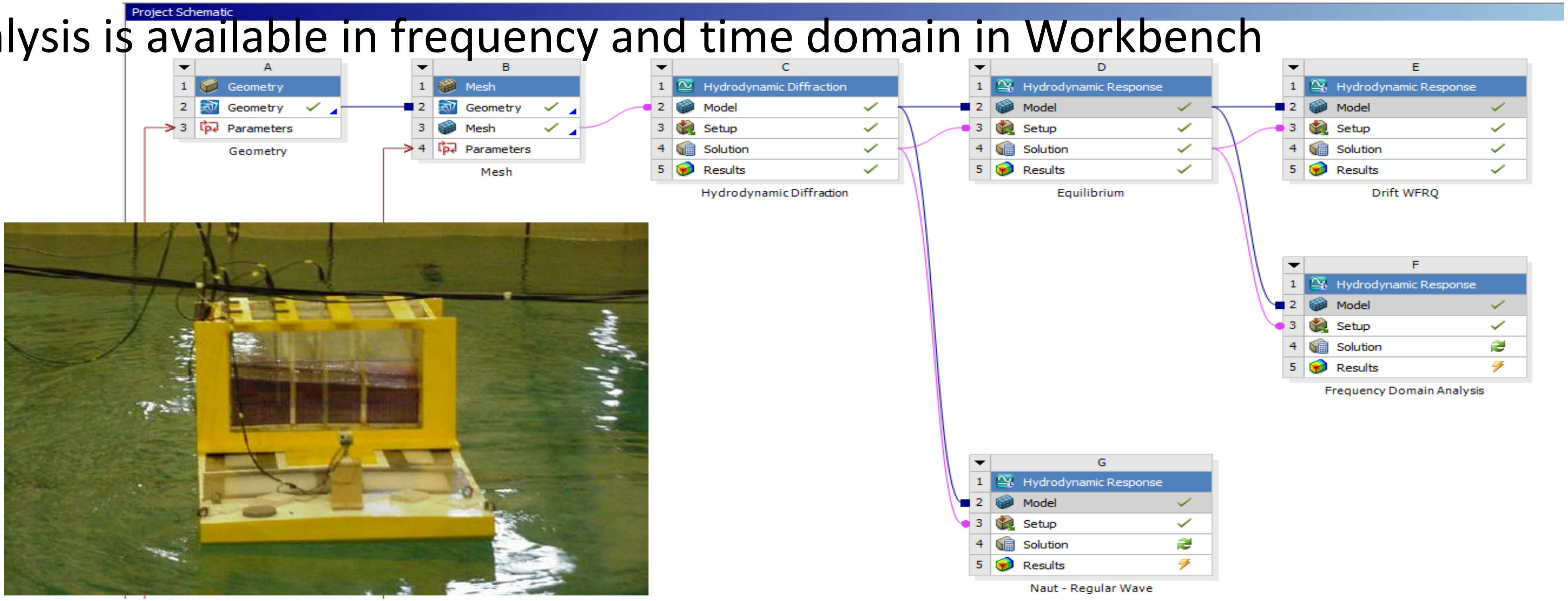
Project/Model Level	Individual Aqwa Specific item
Whole existing Aqwa model data (geometry, connection, constraint, environment)	Each item listed in left column





SA801: Internal tank hydrodynamic coupling

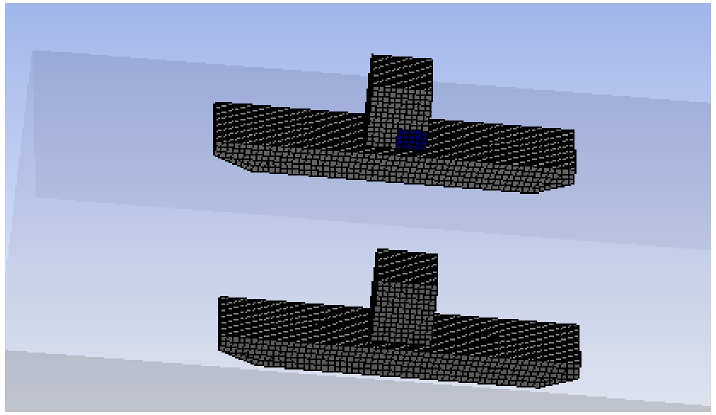
- Coupling effects between liquid motion (sloshing) in the partially filled internal tanks and the attached marine structures
- Hydrostatic and hydrodynamic coupling included
- Analysis is available in frequency and time domain in Workbench

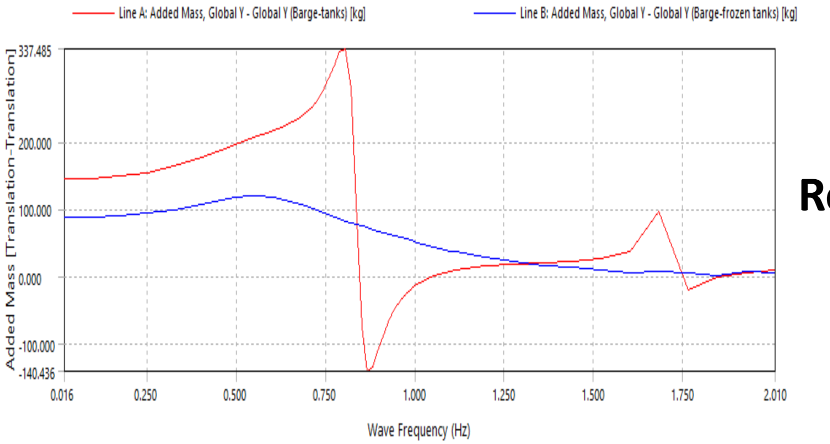


Molin test model

SA801: Internal tank hydrodynamic coupling (example)

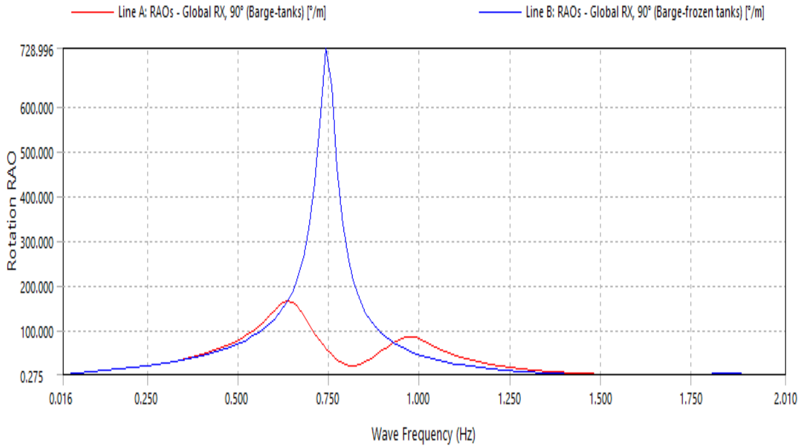
- Two models: (1) with liquid sloshing effects; (2) liquid is frozen (no sloshing)



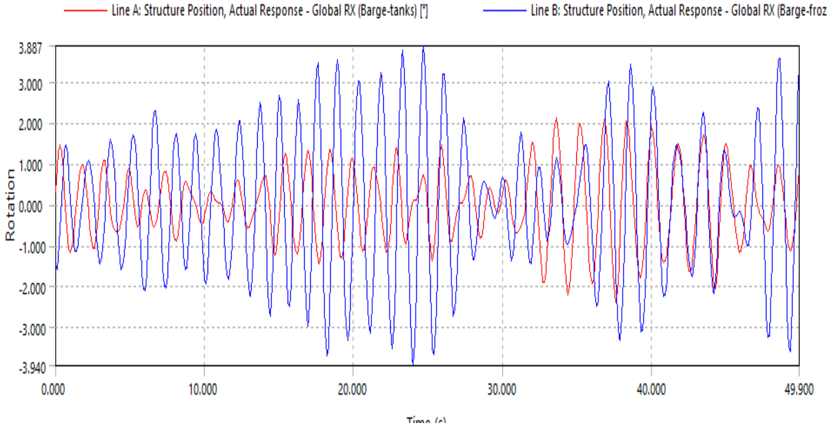


Roll-roll added mass

red line : with sloshing effects
blue line: no sloshing



Roll motion in frequency domain



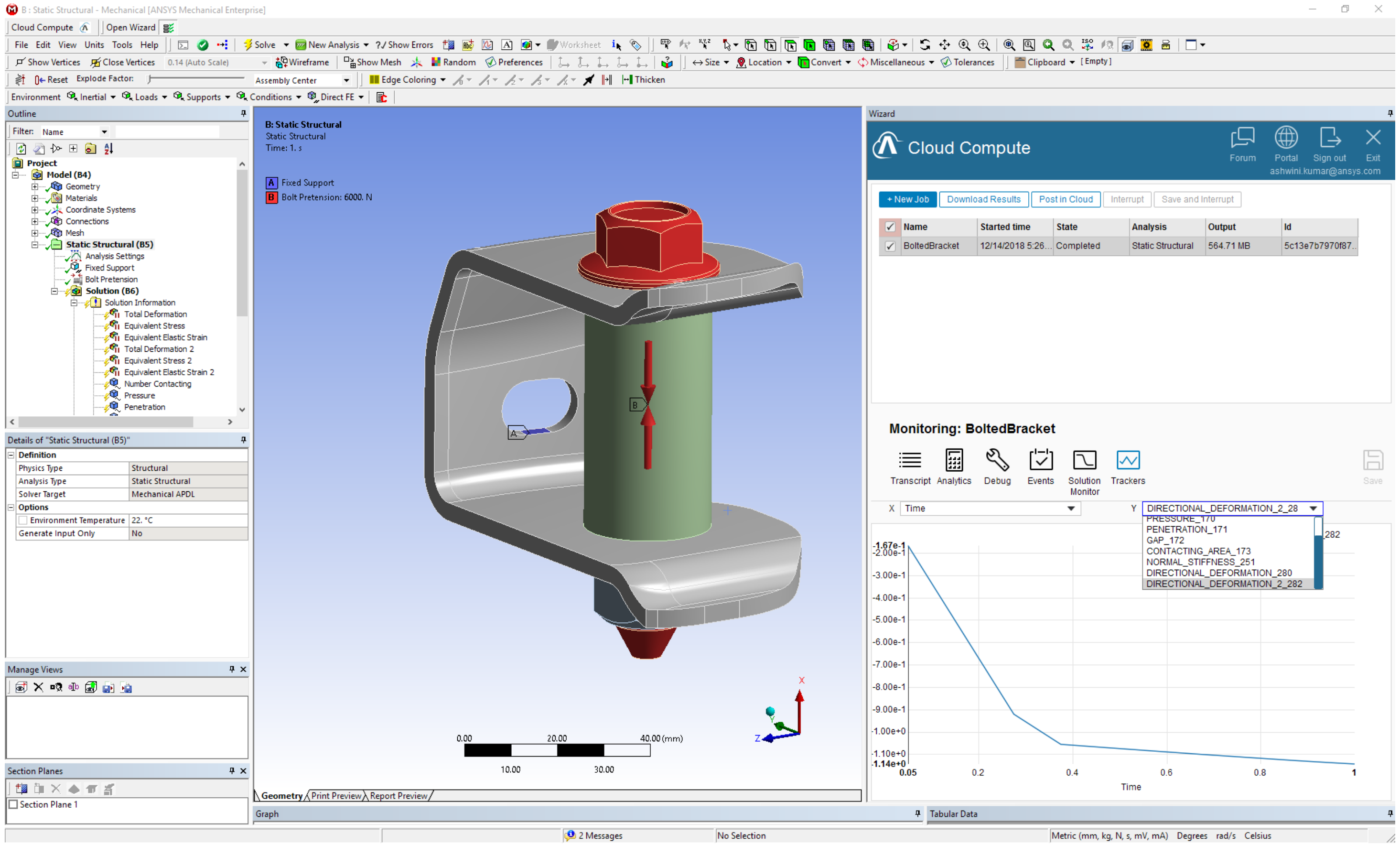
Time history of ship roll motion

ANSYS Cloud

ANSYS Cloud

Cloud-based service that provides easy access to on-demand HPC directly from ANSYS Mechanical & Fluent solver

ANSYS Cloud Compute enables “Solve on Cloud”



Pre — Preprocessing on desktop – geometry prep, meshing, physics, boundary conditions

Solve — Files uploaded to cloud for solve, including automated optimization and DOE

Check — Validate results with remote 3D viewer. Leave heavy weight results in cloud

Post — Download results to desktop for detailed post-processing and data management

Accessing ANSYS Cloud

Register for ANSYS Account

Download ANSYS Cloud Compute ACT App from ANSYS Cloud Portal

Using ANSYS Cloud with ANSYS Mechanical

Install the ANSYS Cloud App

Launch and log-in into the App

Monitor the job from the app and the portal

Post processing in the cloud

Downloading results to your desktop

ANSYS Cloud Forum

Forum to get help and provide feedback

Users can:

- **search the forum for relevant articles**
- **ask a question**
- **post ideas for feature enhancements**